5-1-1977

A static stress analysis of the glass metal truss-type column of the Van De Graaff Accelerator

Rudolf Prechter

A STATIC STRESS ANALYSIS
OF THE
GLASS METAL TRUSS-TYPE COLUMN
OF THE VAN DE GRAAFF ACCELERATOR

by

Rudolf E. Prechter

A Thesis Submitted
in
Partial Fulfillment
of the
Requirements for the Degree of
Master of Science
in
Mechanical Engineering

Approved by:

Prof. Richard B Hetnarski
(Thesis advisor)

Prof. Wayne W. Walter

Prof. Signature Illegible

Prof. Robert M. Desmond
(Department Head)

DEPARTMENT OF MECHANICAL ENGINEERING
COLLEGE OF ENGINEERING
ROCHESTER INSTITUTE OF TECHNOLOGY
ROCHESTER, NEW YORK

G924732

ABSTRACT

A static stress analysis of the glass-metal truss type bridge of a Van de Graaff Accelerator is presented.

The solutions are first computed using conventional methods such as method of joints and the method of virtual work. Secondly, the solutions are found using a computer program, SAP IV, which is a structural analysis program for static and dynamic response of linear systems. The solutions are then compared to each other and appropriate conclusions are presented.

TABLE OF CONTENTS

Page

List of Tables

Page

List of Figures

Page

1. INTRODUCTION

The static stress analysis attempted in this paper involves the Van de Graaff Accelerator Model MP-4. This type of accelerator is used for nuclear studies of elementary particles and is capable of accelerating these particles to potentials as high as 10 million volts. The accelerator consists of a cylindrical pressure vessel which supports a glass-metal truss-type bridge (Fig. 1,2,3). This bridge was designed as a conventional pin-jointed truss except that certain truss members are constructed of alternate layers of glass and metal which are bonded together with an adhesive. Since the tensile strength of the bonded joints has an ultimate value of 1000psi it becomes important to maintain a compressive force in all bonded glass-metal truss members. This compressive force has to be maintained for various operating conditions which include: a) Pressure vessel without internal pressure. b) Pressure vessel with internal pressures of up to 150 psi. c) Internal force of 1000 lbs on the bridge structure under conditions a) and/or b).

Another important factor is the stiffness of the bridge. It is imperative to maintain perfect alignment of certain electronic instruments and magnets and the displacement of these under conditions a), b) and c) must be known precisely. Only then is it possible to align those instruments before actual experiments take place, and to guarantee the alignment for various operating conditions.

10 MV "EMPEROR" TANDEM

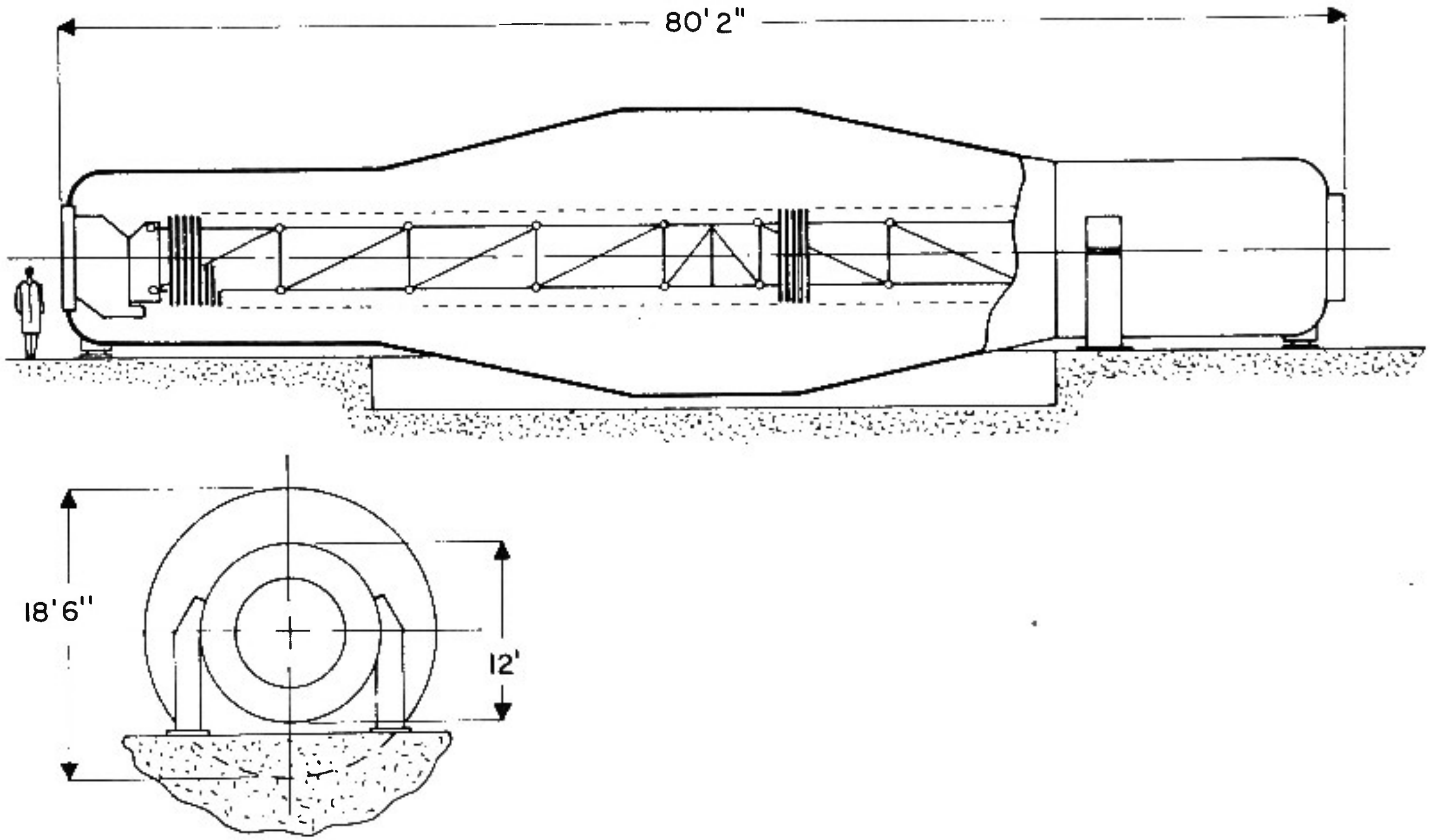
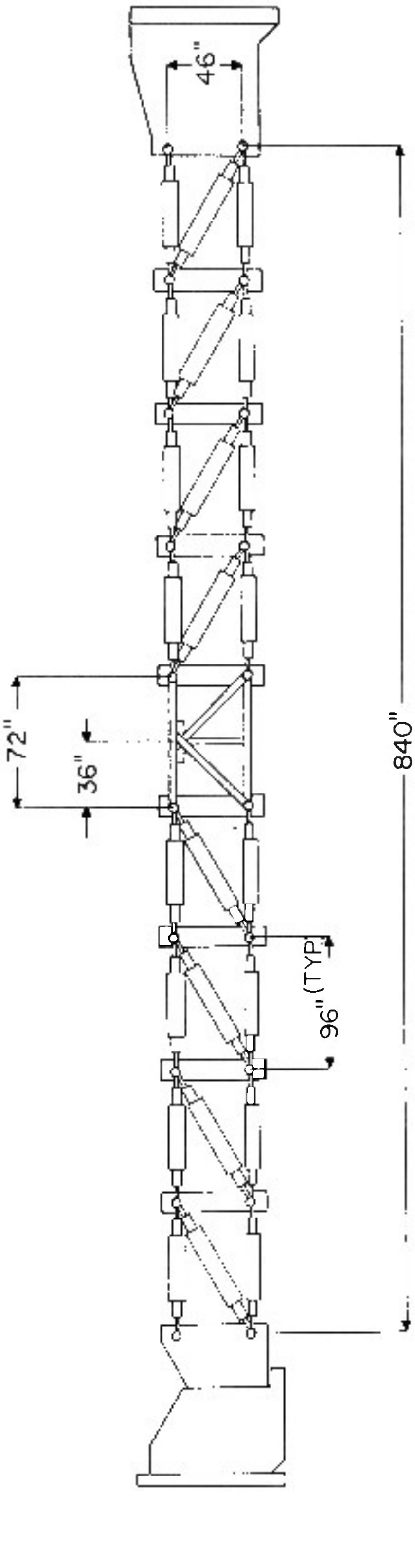
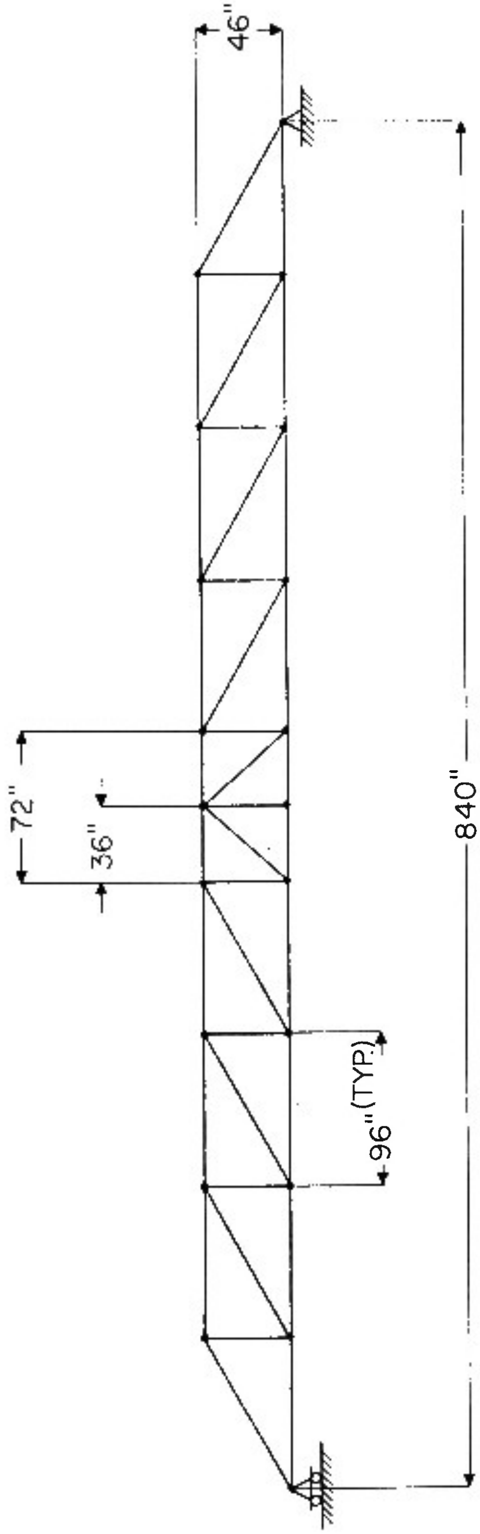


Fig. I



ACTUAL TRUSS

Fig.2



IDEALIZED TRUSS

Fig.3

2. LITERATURE REVIEW

An extensive search was made to find existing studies on this structure. A total of four papers was found which all originated from the manufacturer and which are:

a) Engineering Report No. 359, High Voltage Engineering Corporation, Burlington, Massachusetts.

b) MP Equipment and Stytem Test Report # 288, High Voltage Engineering Corporation, Burlington, Massachusetts.

c) Engineering Report No. 408, High Voltage Engineering Corporation, Burlington, Massachusetts.

d) Mechanical Design of the 20 MeV Emperor Tandem Van de Graaff Accelerator, L.E. Wilson and J. Christofferson, High Voltage Engineering Corporation, Burlington, Massachusetts.

The first paper, Engineering Report No. 359, concerns itself with strain-gauge measurements on the first existing glass-metal truss-type bridge of the Van de Graaff Accelerator at High Voltage Engineering Corporation.

The second paper, MP Equipment and System Test Report No. 288, concerns itself with measurements on the structure during pressurization of the pressure vessel.

The third paper, Engineering Report No. 408, concerns itself with scale model tests on the MP column structure.

The fourth paper, Mechanical Design of the 20 MeV Emperor Tandem Van de Graaff Accelerator, concerns itself with the overall general mechanical design of the accelerator and gives no specific details as to stress and strain.

All four papers concern themselves with measurements on an accelerator which is similar to the model MP-4 under consideration in this paper. However, the loading conditions are different and the results as published in these four papers cannot directly be applied to the model MP-4.

Certain physical constants, such as length, area, modulus of elasticity, etc. are important and are used for the calculations in this paper. These constants are listed in TABLES 25 and 26.

3. OBJECT

Since a review of the existing literature shows that theoretical calculations have not been published or are not available a complete stress analysis is desirable. In this paper, therefore, a detailed analysis of forces developed in each component of the structure as well as vertical displacements of certain points in the structure is undertaken.

These calculations are made for the loading and operating conditions of the model MP-4 accelerator which is located at the University of Rochester, Rochester, N.Y.

4. SCOPE

a) Computations of forces in bridge members treating the bridge as an ideal, simply supported Howe-truss. Method of joints is used for calculations.

b) Computation of force necessary to keep the bridge-structure under compression.

c) Computation of forces as in part 4b, but including the internal force of 1000 lbs. generated under certain operating conditions.

d) Computation of forces as in part 4c, but including internal pressure of 150 psi in pressure vessel.

e) Computation of displacement at the mid-point of the bridge-structure with pressure vessel under internal pressure of 150 psi and with 1000 lbs. internal force.

f) Computation of displacement at the mid-point of the structure without internal pressure.

g) Computation of forces and displacements under conditions as in parts 4a, 4b, 4c, 4d, 4e, 4f using the structure analysis program SAP IV which is based on finite element methods.

h) Comparison of forces and displacements as calculated by method of joints and method of virtual work versus SAP IV.

5. THEORY

A. For the computation of forces in the structural members of the truss-type bridge the method of joints is used. To use this technique a free body diagram of any joint in the truss must be drawn, provided no more than two unknown forces act at that joint. After forces are calculated at that joint, the other unknown forces can be calculated by proceeding from one joint to another.

The necessary equations of equilibrium are:

$$\Sigma F_x = 0$$

$$\Sigma F_y = 0$$

$$\Sigma M_a = 0$$

Other important assumptions are:

a) Each truss is assumed to be composed of rigid members all lying in one plane.

b) Forces are transmitted from one member to another through frictionless pins.

c) All loads are assumed to act only at the joints.

Example:

1) <u>Free-body Diagram</u>:

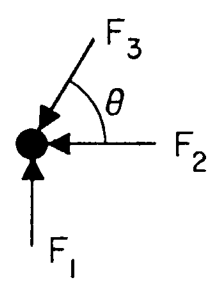


Fig.4

2) <u>Equations of Equilibrium</u>

$$\Sigma F_x = 0 = -F_2 - F_3 \cos \Theta$$

$$\Sigma F_y = 0 = F_1 - F_3 \sin \Theta$$

3) Solution:

$$F_2 = -F_3 \cos \Theta$$

$$F_3 = F_1/\sin \Theta$$

B. For the computation of displacements the method of complementary virtual work is used.

The principle involved is:

The complementary virtual work δW_E done by an external virtual force system under the actual deformation of a structure is equal to the complementary work δU done by the virtual stresses under the actual strains.[1]

The following discussion closely follows the book, Elementary Theory of Structures by Yuan-Yu Hsien.[2] This principle provides an extremely useful means for calculating unknown displacements.

Consider a structure, Fig.5 , subject to forces P_1 and P_2 which move distances Δ_1, Δ_2.

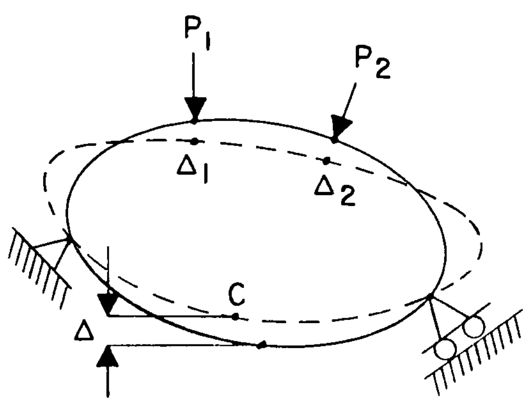


Fig.5

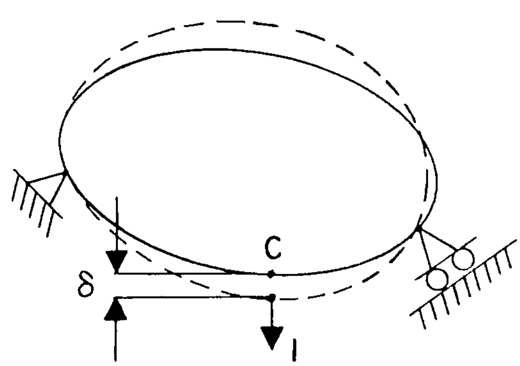


Fig.6

---

[1] Energy Principles in Structural Mechanics, Theodore R. Tauchert, McGray-Hill Book Company, N.Y. 1974

[2] Elementary Theory of Structures, Yuan-Yu Hsien, Prentice-Hall, Inc. Englewood Cliffs, N.J. 1970.

In order to find the deformation at any point of the structure, C for example, we visualize the same structure, Fig.6 , with all the actual loads removed but a virtual load of unity being applied at point C along the desired deflection which moves distance δ. A typical element of Fig.5 of length L which is subject to internal forces S moves a distance dL (Fig.7).

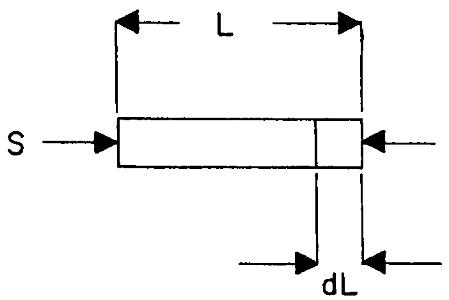


Fig.7

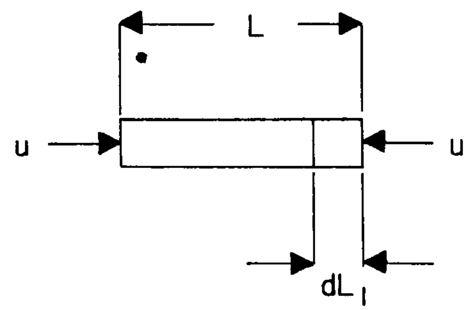


Fig.8

The same element, subject to internal forces, u, has a change in length of dL_1 (Fig.8). However, the external work done by the applied loads must equal the internal strain energy and we get for Fig. 5 and Fig.7 that:

$$1/2P_1\Delta_1 + 1/2P_2\Delta_2 = 1/2\Sigma SdL \qquad (1)$$

And for Fig. 6 and Fig.8 :

$$1/2(1)(\delta) = 1/2\,\Sigma\, udL_1 \qquad (2)$$

If the case in Fig. 6 exists first and the actual loads P_1 and P_2 are then applied to it, one can equate the total work done and the total strain energy restored during this period.

$$(1/2)(1)(\delta) + 1/2P_1\Delta_1 + 1/2P_2\Delta_2 + (1)\Delta$$
$$= 1/2\,\Sigma\, udL_1 + 1/2\,\Sigma SdL + \Sigma\, udL \qquad (3)$$

Since the strain energy and work done must be the same whether the loads are applied together or separately, we can subtract the sum of equations (1) and (2) from equation (3) to get:

$$1 \cdot \Delta = \Sigma\, udL \qquad (4)$$

For the case of any joint of a loaded truss each member of the truss can be considered as an element. The term dL in equation (4) is then the shortening or lengthening of a bar due to applied loads and can be expressed as:

$$dL = \frac{SL}{AE} \qquad (5)$$

Equation (4) then becomes:

$$1 \cdot \Delta = \Sigma\, \frac{SuL}{AE} \qquad (6)$$

where:

S = internal force in any member due to actual loads

u = the internal force in the same member due to a fictitious unit load at the point where the deflection is sought, acting along the desired direction

L = length of member

A = cross sectional area of the member

E = modulus of elasticity of the member.

Example: Find the vertical deflection of joint 3 of the loaded truss shown in Fig. 9.

Assume: $L(ft)/A(in^2) = 1$, $E = 30{,}000\ Kips/in^2$

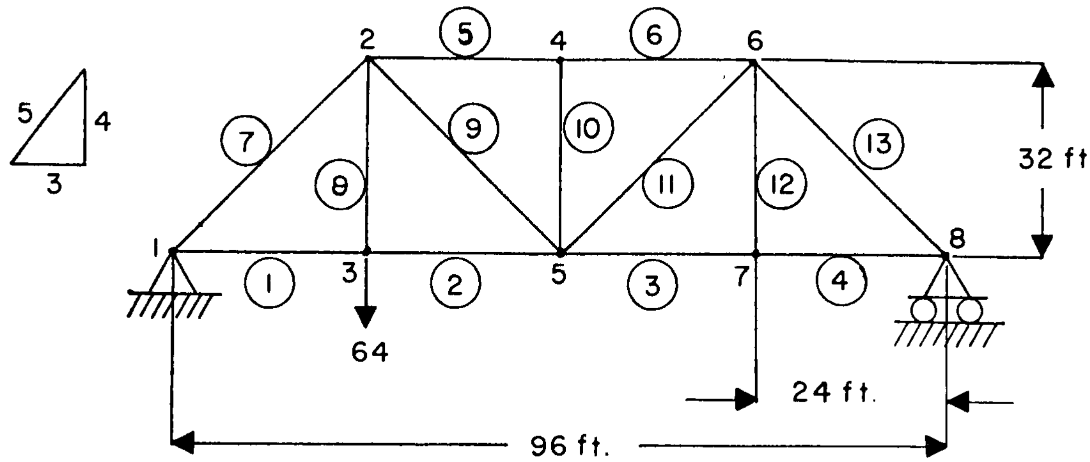


Fig.9

I)Solution for S:

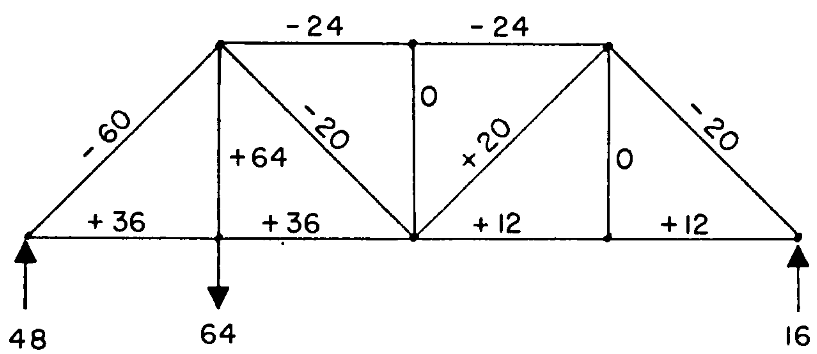


Fig.10

II) Solution for u:

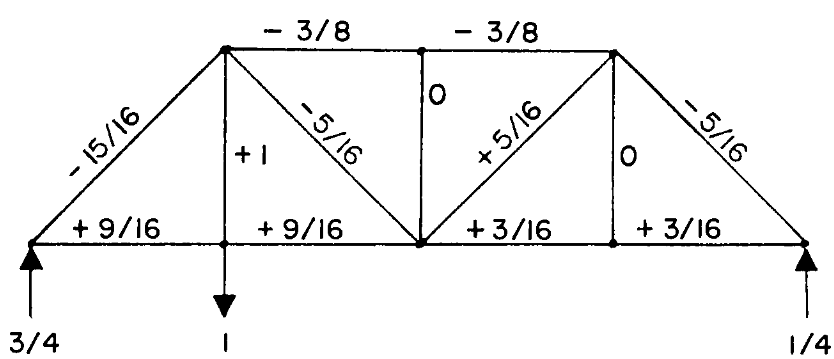


Fig.II

TABLE 1

III) Solution for Δ:

Member	L/A(ft/in^2)	S(Kips)	u	$\frac{SuL}{A}(\frac{ft\text{-}Kips}{in^2})$
1	1	+36	+9/16	+20.25
2	1	+36	+9/16	+20.25
3	1	+12	+3/16	+ 2.25
4	1	+12	+3/16	+ 2.25
5	1	-24	-3/8	+ 9.0
6	1	-24	-3/8	+ 9.0
7	1	-60	-15/16	+56.25
8	1	+64	+ 1	+64.0
9	1	-20	-5/16	+ 6.25
10	1	0	0	0
11	1	+20	+5/16	+ 6.25
12	1	0	0	0
13	1	-20	-5/16	+ 6.25

$\Sigma \quad + 202.0$

$$\Delta = \Sigma \frac{SuL}{AE} = \frac{+202.0}{30\ 000} = +\ 0.00673 \text{ ft}$$

C. SAP IV, a structural analysis program for static and dynamic response of linear systems, is used for the analysis of forces and displacements. This program is based on finite element principles and therefore a discussion of the theory of finite elements as applied to axial members of a truss is included here. This discussion closely follows the book, Introduction to Finite Element Analysis by H.C. Martin and G.F. Carey[3].

Consider the pin-jointed truss in Fig. 12.

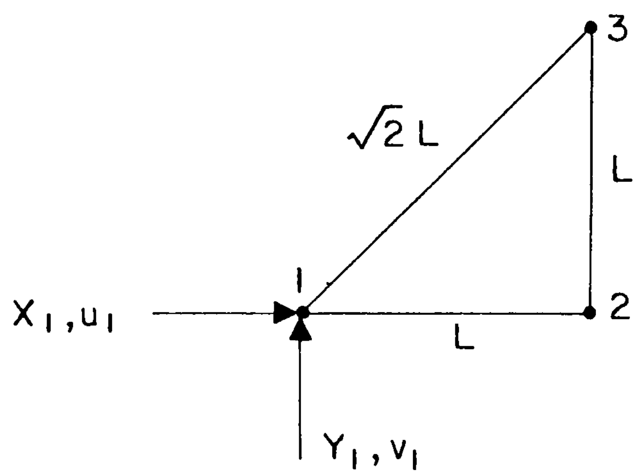


Fig.12

Assume forces and displacements to be relevant for the nodal points 1, 2, and 3. Since there are six possible displacements, the total structure has six degrees of freedom. For a solution, we have to relate these six displacements to the six nodel forces. If we assume the structure to be linearly elastic displacements must be directly proportional to the

---

3) Introduction to Finite Element Analysis, Harold C. Martin, Graham F. Carey, McGraw-Hill Book Company, N.Y. 1973.

forces. If we express the force system with displacements $u_1 \neq 0$, and $u_2 = u_3 = v_1 = v_2 = v_3 = 0$, we get:

$$X_1 = k_{11}^{xx} u_1 \tag{7}$$

If we express the force system for each degree of freedom and collect them in a single matrix equation we get: (8)

$$\begin{bmatrix} X_1 \\ Y_1 \\ X_2 \\ Y_2 \\ X_3 \\ Y_3 \end{bmatrix} = \begin{bmatrix} k_{11}^{xx} & k_{11}^{xy} & k_{12}^{xx} & k_{12}^{xy} & k_{13}^{xx} & k_{13}^{xy} \\ k_{11}^{yx} & k_{11}^{yy} & k_{12}^{yx} & k_{12}^{yy} & k_{13}^{yx} & k_{13}^{yy} \\ k_{21}^{xx} & k_{21}^{xy} & k_{22}^{xx} & k_{22}^{xy} & k_{23}^{xx} & k_{23}^{xy} \\ k_{21}^{yx} & k_{21}^{yy} & k_{22}^{yx} & k_{22}^{yy} & k_{23}^{yx} & k_{23}^{yy} \\ k_{31}^{xx} & k_{31}^{xy} & k_{32}^{xx} & k_{32}^{xy} & k_{33}^{xx} & k_{33}^{xy} \\ k_{31}^{yx} & k_{31}^{yy} & k_{32}^{yx} & k_{32}^{yy} & k_{33}^{yx} & k_{33}^{yy} \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix}$$

This equation relates all possible nodal forces to all possible nodal displacements. The coefficients k_{ij}^{mn} equal the force at node i acting in the direction which is required to sustain a unit displacement at node j acting in the n direction.

The formation of this stiffness equation represents the crucial step in any finite element method of analysis.

If we now impose displacements on the truss of Fig. 12 such as: $u_2 = v_2 = u_3 = v_3 = 0$

then the truss is supported against rigid-body displacement. Under these conditions, forces X_2, Y_2, X_3 and Y_3 become unknown reactions, while X_1 and Y_1 can be considered as possible applied loadings on the system. Equation (8) can then be written in the partitioned form:

$$\begin{bmatrix} X_1 \\ Y_1 \\ X_2 \\ Y_2 \\ X_3 \\ Y_3 \end{bmatrix} = \begin{bmatrix} K_{\alpha\alpha} & K_{\alpha\beta} \\ K_{\alpha\beta}^{T} & K_{\beta\beta} \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 = 0 \\ v_2 = 0 \\ u_3 = 0 \\ v_3 = 0 \end{bmatrix} . \tag{9}$$

The unknowns in this equation are the displacements u_1 and v_1. These are found by matrix multiplication. This gives two equations:

$$\begin{bmatrix} X_1 \\ Y_1 \end{bmatrix} = K_{\alpha\alpha} \begin{bmatrix} u_1 \\ v_1 \end{bmatrix} \tag{10}$$

$$\begin{bmatrix} X_2 \\ Y_2 \\ X_3 \\ Y_3 \end{bmatrix} = K_{\alpha\beta}^{T} \begin{bmatrix} u_1 \\ v_1 \end{bmatrix} \tag{11}$$

If we invert $K_{\alpha\alpha}$ of equation (10) we get the unknown displacements.

$$\begin{bmatrix} u_1 \\ v_1 \end{bmatrix} = K_{\alpha\alpha}^{-1} \begin{bmatrix} X_1 \\ Y_1 \end{bmatrix} \tag{12}$$

The unknown forces can then be obtained by substituting equation (12) into equation (11).

$$\begin{bmatrix} X_2 \\ Y_2 \\ X_3 \\ Y_3 \end{bmatrix} = K_{\alpha\beta}^{T} \quad K_{\alpha\alpha} \begin{bmatrix} X_1 \\ Y_1 \end{bmatrix} \tag{13}$$

It is clear from the foregoing discussion that the basic unknowns can be found if the individual stiffness matrices of the elements can be determined.

In the case of the axial elements in a truss, this matrix is as follows:

$$\begin{bmatrix} \bar{X}_i \\ \bar{X}_j \end{bmatrix} = \frac{AE}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} \bar{u}_i \\ \bar{u}_j \end{bmatrix} \tag{14}$$

This matrix equation refers to local coordinates of the element. For the element in system coordinates we get:

$$\begin{bmatrix} X_1 \\ Y_1 \\ X_2 \\ Y_2 \end{bmatrix} = \frac{AE}{L} \begin{bmatrix} \lambda^2 & \lambda\mu & -\lambda^2 & -\lambda\mu \\ \lambda\mu & \mu^2 & -\lambda\mu & -\mu^2 \\ -\lambda^2 & -\lambda\mu & \lambda^2 & \lambda\mu \\ -\lambda\mu & -\mu^2 & \lambda\mu & \mu^2 \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{bmatrix} \tag{15}$$

where $\lambda = \cos\Theta$ and $\mu = \sin\Theta$, are the direction cosines of the individual elements.

In order to form the stiffness equation for the entire truss we superimpose the individual element stiffness matrices and immediately get the gross stiffness matrix. With the gross stiffness matrix known we can now go back to equations (9)(10)(11)(12)(13) and solve for the basic unknowns.

A summary of the principle steps involved includes:

1) Definition of the idealized problem as an assemblage of elements.
2) Derivation of individual element stiffness matrices in local coordinates.
3) Calculation of the element stiffness matrices and transformation to system coordinates.
4) Superposition of element stiffness matrices to obtain the gross assemblage stiffness equation.
5) Solution for unknown displacements.

For a general outline and organization of the SAP IV computer program[4)] see Appendix A, Page 93.

For data input to SAP IV see Appendix B, Page 115.

For a sample printed output of SAP IV see Appendix C, Page 128.

---

4) SAP IV, A structural analysis program for static and dynamic response, Klaus-Jürgen Bathe, Edward L. Wilson, Fred E. Peterson, College of Engineering, University of California, Berkeley, California.

VI. COMPUTATION OF FORCES IN TRUSS MEMBERS

A. Without Compressive Spring Force

For the calculation of these forces, we proceed as follows:

1) Draw diagram of idealized truss with load and reactive forces. (Fig. 13)
2) Calculate reaction forces
3) Calculate, by method of joints, forces in individual members.
4) CHECK ON CALCULATIONS.

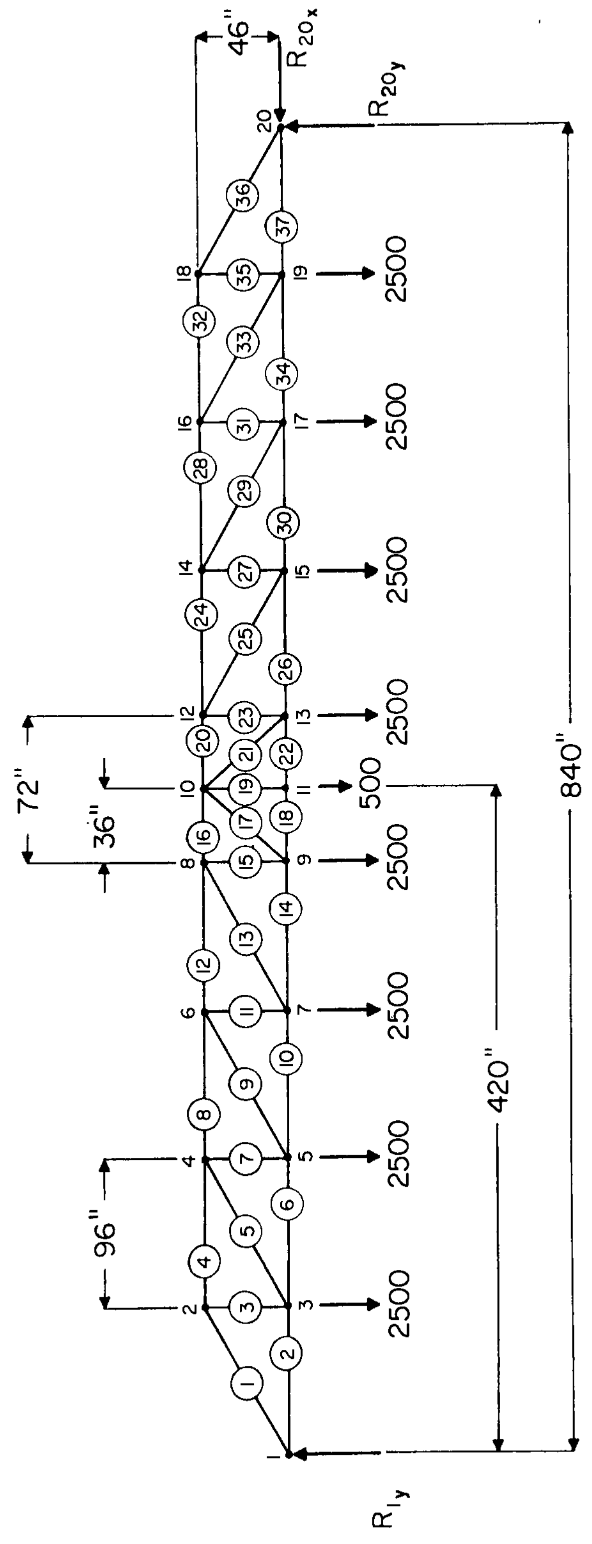
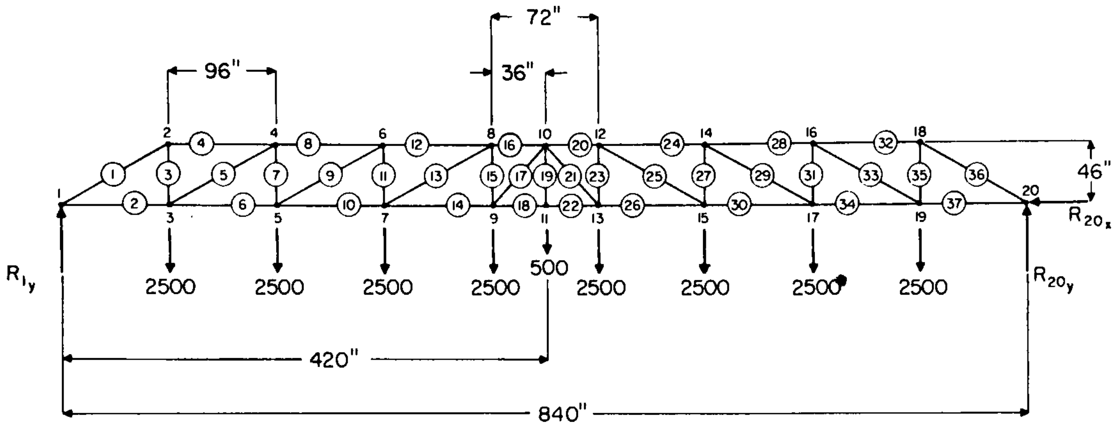


Fig.I3

Calculation of Reaction Forces



$$\Sigma F_x = 0 = R_{20x}$$

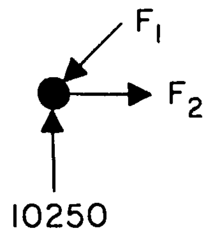
$$\Sigma F_y = 0 = R_{1y} + R_{20y} - 20500$$

By reasons of symmetry: $R_{1y} = 10250$

$R_{20y} = 10250$

Calculation of Truss-Member Forces

1)



$$\Sigma F_x = 0 = F_2 - F_1 \cos 25.6$$

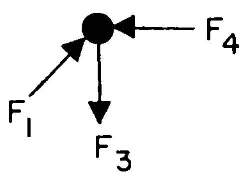
$$\Sigma F_y = 0 = 10250 - F_1 \sin 25.6$$

$$F_1 = 23720.3$$

$$F_2 = 21391.3$$

---

2)



$$\Sigma F_x = 0 = F_1 \cos 25.6 - F_4$$

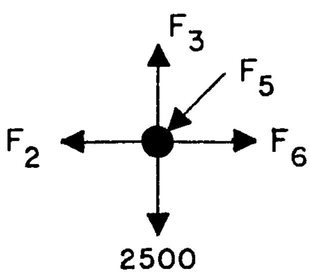
$$\Sigma F_y = 0 = F_1 \sin 25.6 - F_3$$

$$F_3 = 10250.0$$

$$F_4 = 21391.3$$

---

3)



$$\Sigma F_x = 0 = -F_2 + F_6 - F_5 \cos 25.6$$

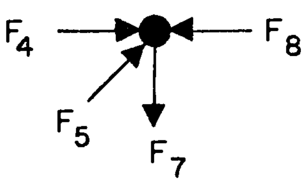
$$\Sigma F_y = 0 = F_3 - F_5 \sin 25.6 - 2500$$

$$F_5 = 17934.8$$

$$F_6 = 37565.2$$

---

4)



$$\Sigma F_x = 0 = F_4 - F_8 + F_5 \cos 25.6$$

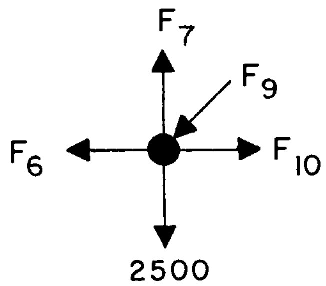
$$\Sigma F_y = 0 = F_5 \sin 25.6$$

$$F_7 = 7750$$

$$F_8 = 37565.2$$

---

5)



$$\Sigma F_x = 0 = F_{10} - F_6 - F_9 \cos 25.6$$

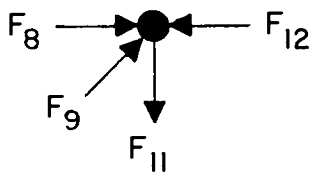
$$\Sigma F_y = 0 = F_7 - 2500 - F_9 \sin 25.6$$

$$F_9 = 12149.4$$

$$F_{10} = 48521.7$$

---

6)



$$\Sigma F_x = 0 = F_8 + F_9 \cos 25.6 - F_{12}$$

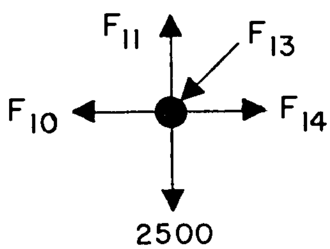
$$\Sigma F_y = 0 = F_9 \sin 25.6 - F_{11}$$

$$F_{11} = 5250.0$$

$$F_{12} = 48521.7$$

---

7)



$$\Sigma F_x = 0 = F_{14} - F_{10} - F_{13} \cos 25.6$$

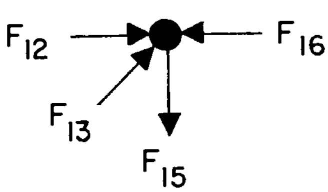
$$\Sigma F_y = 0 = F_{11} - 2500 - F_{13} \sin 25.6$$

$$F_{13} = 6363.9$$

$$F_{14} = 54260.9$$

---

8)



$$\Sigma F_x = 0 = F_{12} - F_{16} + F_{13} \cos 25.6$$

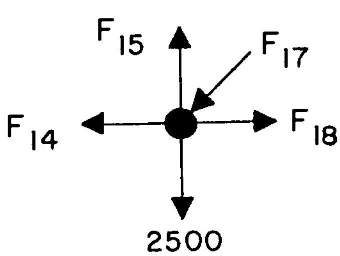
$$\Sigma F_y = 0 = F_{13} \sin 25 - F_{15}$$

$$F_{15} = 2750.0$$

$$F_{16} = 54260.9$$

---

9)



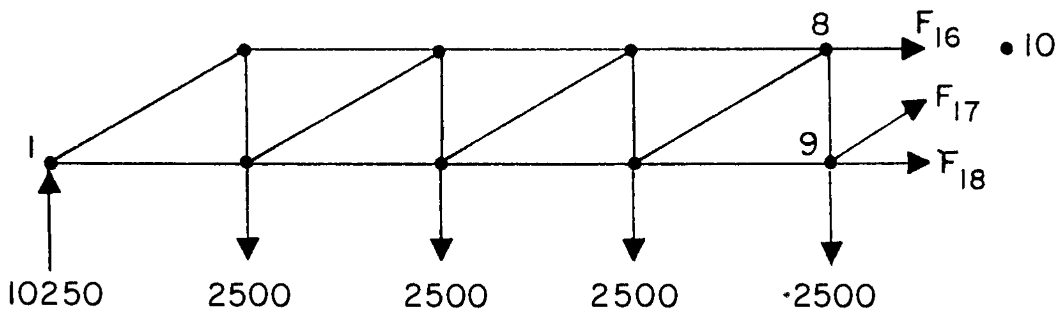
$$\Sigma F_x = 0 = F_{18} - F_{14} - F_{17} \cos 51.9$$

$$\Sigma F_y = 0 = F_{15} - 2500 - F_{17} \sin 51.9$$

$$F_{17} = 317.5$$

$$F_{18} = 54456.5$$

CHECK:



$$\Sigma M_9 = F_{16}(46) + (10250)(384) - (7500)(192) = 0$$

$$F_{16} = -54260.9$$

$$\Sigma M_{10} = (10250)(420) - (10\ 000)(180) - F_{18}(46)$$

$$F_{18} = 54456.5$$

$$\Sigma M_8 = (10250)(384) - (7500)(192) - (54456.52174)(46) - F_{17}(28.350)$$

$$F_{17} = -317.5$$

TABLE 2

Truss Member	Calculated Force	Force Direction
Lower 2, 37	21391.3	Tension
6, 34	37565.2	
10, 30	48521.7	
14, 26	54260.9	
18, 22	54456.5	
Upper 4, 32	- 21391.3	Compression
8, 28	- 37565.2	
12, 24	- 48521.7	
16, 20	- 54260.9	
Diag. 1, 36	- 23720.3	
5, 33	- 17934.8	
9, 29	- 12149.4	
13, 25	- 6363.9	
17, 21	- 317.5	
Vert. 3, 35	10250.0	Tension
7, 31	7750.0	
11, 27	5250.0	
15, 23	2750.0	
19,	500.0	

These calculations show that the highest tensile forces are developed in the lower members of the truss, which is as one would expect.

In order to put the lower members under compression a minimum spring force of 55000 lbs. would be required. If a margin of 5000 lbs is included, we arrive at a force of 60000 lbs. The actual spring force is of the order of 72 800 lbs. which leaves a margin of 17 800 lbs. This high force of 72 8000 lbs. is due to the fact that much higher loads were anticipated in the original design.

B. Computation of Forces with Spring Force of 72 800 lbs.

We proceed as follows:

1) Draw diagram of idealized truss with loads, spring force of 72 800 lbs. and reaction forces (Fig. 14)
2) Calculate reaction forces
3) Calculate, by method of joints, forces in individual members
4) Check on calculations.

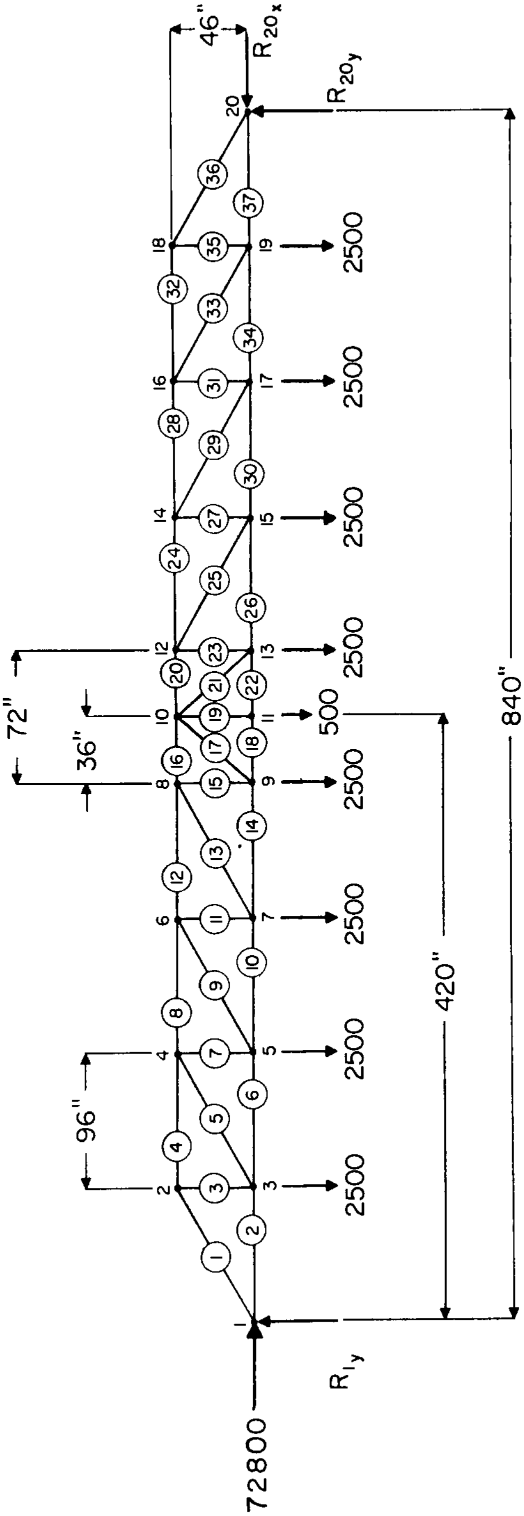
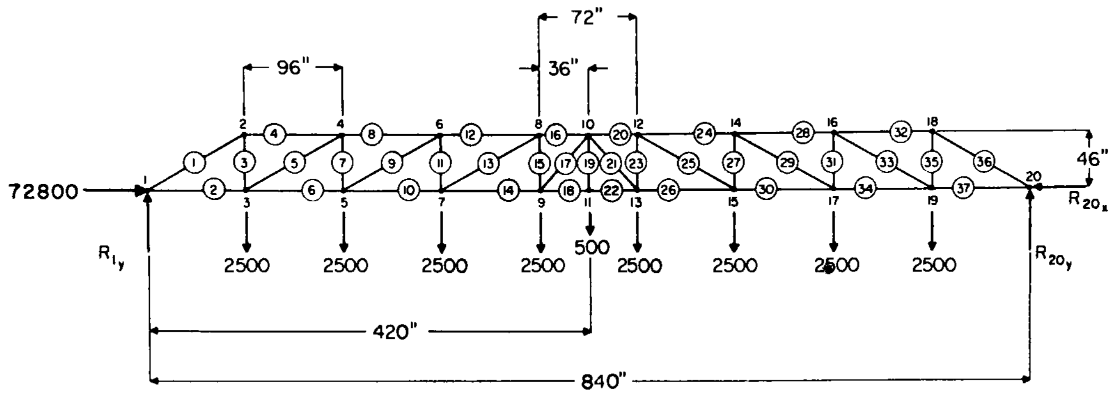


Fig. 14

Computation of Reaction Forces



$$\Sigma F_x = 0 = 72800 - R_{20x}$$

$$R_{20x} = 72800$$

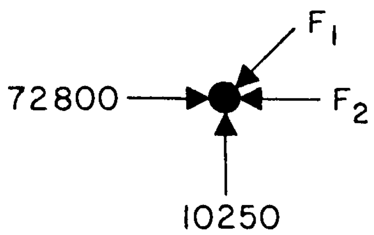
$$\Sigma F_y = 0 = R_{1y} + R_{20y} - 20500$$

By reasons of symmetry: $R_{1y} = 10250.0$

$R_{20y} = 10250.0$

Computation of Truss-Member Forces

1)



$$\Sigma F_x = 0 = 72800 - F_2 - F_1 \cos 25.6$$

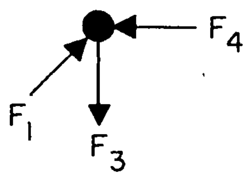
$$\Sigma F_y = 0 = 10250 - F_1 \sin 25.6$$

$$F_1 = 23720.3$$

$$F_2 = 51408.7$$

---

2)



$$\Sigma F_x = 0 = F_1 \cos 25.6 - F_4$$

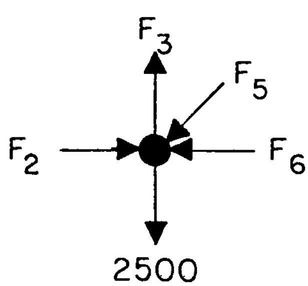
$$\Sigma F_y = 0 = F_1 \sin 25.6 - F_3$$

$$F_3 = 10250.0$$

$$F_4 = 21391.3$$

---

3)



$$\Sigma F_x = 0 = F_2 - F_6 - F_5 \cos 25.6$$

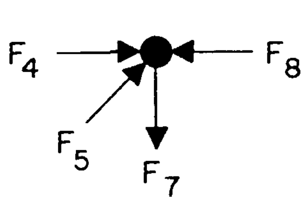
$$\Sigma F_y = 0 = F_3 - 2500 - F_5 \sin 25.6$$

$$F_5 = 17934.8$$

$$F_6 = 35234.8$$

---

4)



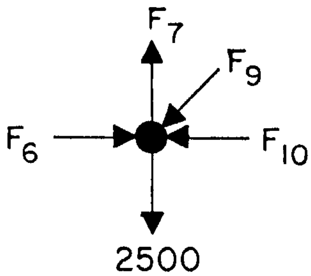
$$\Sigma F_x = 0 = F_4 + F_5 \cos 25.6 - F_8$$

$$\Sigma F_y = 0 = F_5 \sin 25.6 - F_7$$

$$F_7 = 7750.0$$

$$F_8 = 37565.2$$

5)



$\Sigma F_x = 0 = F_6 - F_{10} - F_9 \cos 25.6$

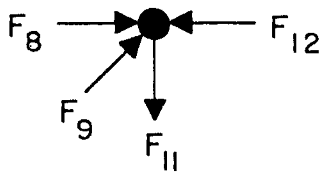
$\Sigma F_y = 0 = F_7 - 2500 - F_9 \sin 25.6$

$F_9 = 12149.4$

$F_{10} = 24278.3$

---

6)



$\Sigma F_x = 0 = F_8 + F_9 \cos 25.6 - F_{12}$

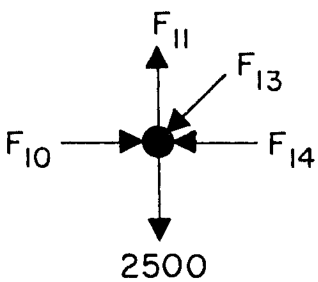
$\Sigma F_y = 0 = F_9 \sin 25.6 - F_{II}$

$F_{II} = 5250.0$

$F_{12} = 48521.7$

---

7)



$\Sigma F_x = 0 = F_{10} - F_{13} \cos 25.6 - F_{14}$

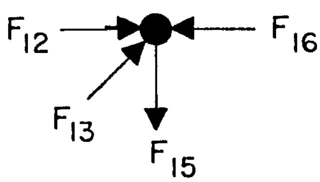
$\Sigma F_y = 0 = F_{11} - F_{13} \sin 25.6 - 2500$

$F_{13} = 6363.9$

$F_{14} = 18539.3$

---

8)



$\Sigma F_x = 0 = F_{12} + F_{13} \cos 25.6 - F_{16}$

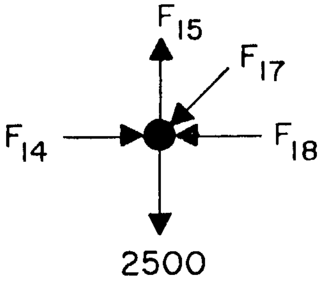
$\Sigma F_y = 0 = F_{13} \sin 25.6 - F_{15}$

$F_{15} = 2750.0$

$F_{16} = 54260.9$

---

9)



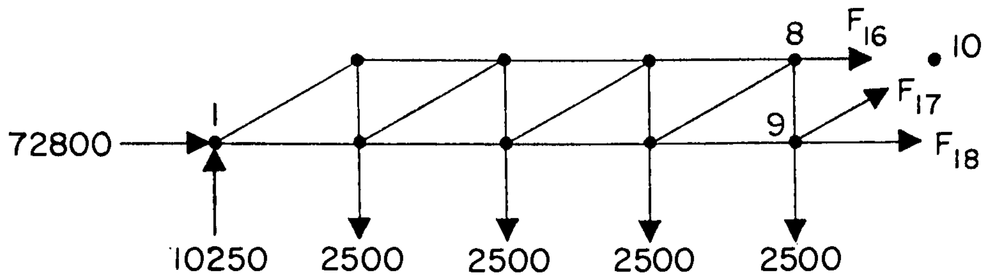
$\Sigma F_x = 0 = F_{14} - F_{17} \cos 51.9 - F_{18}$

$\Sigma F_y = 0 = F_{15} - F_{17} \sin 51.9 - 2500$

$F_{17} = 317.5$

$F_{18} = 18343.6$

CHECK:



$$\Sigma M_9 = F_{16}(46) + (10250)(384) - (7500)(192) = 0$$

$$F_{16} = -\ 54260.9$$

$$\Sigma M_{10} = (10250)(420) - (10000)(180) - (72800)(46) = F_{18}(46)$$

$$F_{18} = -\ 18343.5$$

$$\Sigma M_8 = (10250)(384) - (7500)(192) - (72800)(46) + (+18343.5)(46) - F_{17}(28.4)$$

$$F_{17} = -\ 317.5$$

TABLE 3

Truss-Member		Calculated Force	Force-Direction
Lower	2,37	- 51 408.7	Compression
	6,34	- 35 234.8	
	10,30	- 24 278.3	
	14,26	- 18 539.3	
	18,22	- 18 343.6	
Upper	4,32	- 21 391.3	
	8,28	- 37 565.2	
	12,24	- 48 521.7	
	16,20	- 54 260.9	
Diagonal	1,36	- 23 720.3	
	5,33	- 17 934.8	
	9,29	- 12 149.4	
	13,25	- 6 363.9	
	17,21	- 317.5	
Vertical	3,35	10 250.0	Tension
	7,31	7 750.0	
	11,27	5 250.0	
	15,23	2 750.0	
	19	500.0	

These calculations show that all lower, upper and diagonal members are now under compression while all vertical members are under tension.

The lowest compressive force in a bonded truss-member is of the order of 6400 lbs. This is a more than adequate margin of safety which allows additional loading of the structure in the future.

The highest compressive force, 51 500 lbs, which corresponds to 805psi, is far below the compressive strengths of 166 000psi.

C. Computation of Forces in Truss-Members with a Spring-Force of 72 800 lbs and Internal Force of 1000 lbs

1) Draw diagram of idealized truss (Fig. 15)
2) Calculate reaction forces
3) Calculate, by method of joints, forces in individual members.
4) Check on calculations.

The check in this case is automatic since calculations are done firstly from the right side to the center and secondly from the left side to the center. If forces calculated in the center agree, then the results are automatically proven.

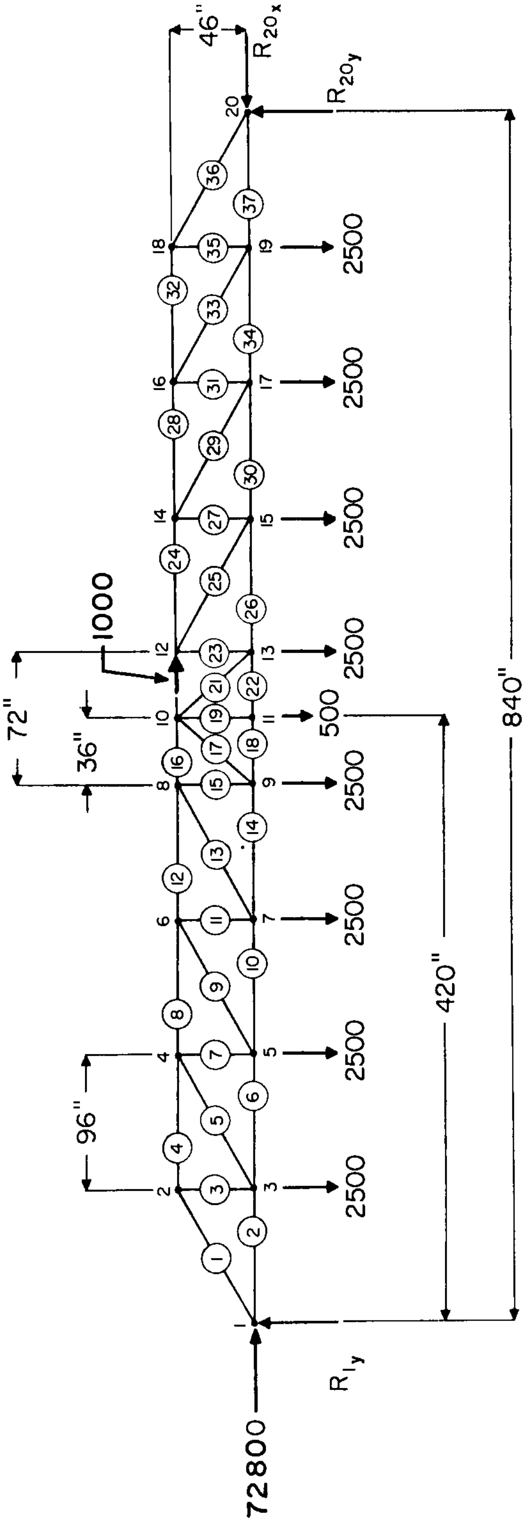
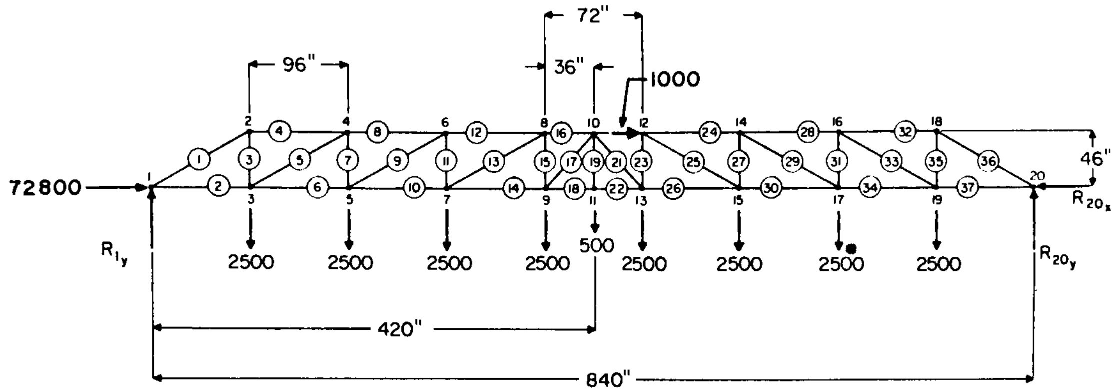

72800
R_{1y}
R_{20x}
R_{20y}
46"
96"
36"
72"
1000
420"
840"
2500
2500
2500
2500
2500
500
2500
2500
2500
2500
2500

Fig. 15

Calculation of Reaction Forces



$$\Sigma M_{20} = 0 = R_{1y}(840)+(1000)(46)-(20500)(420)$$

$$R_{1y} = 10\ 195.2$$

$$\Sigma M_{1} = 0 = (20500)(420)+(1000)(46) - R_{20y}(840)$$

$$R_{20y} = 10\ 304.8$$

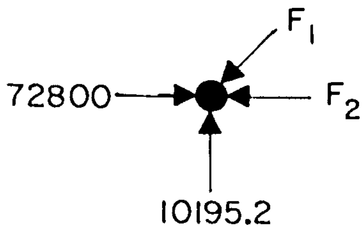
$$\Sigma M_{12} = 0 = R_{1y}(456) + R_{20x}(46)-(72800)(46)-(20500)(36) - R_{20y}(384)$$

$$R_{20x} = (3\ 348\ 800 + 738\ 000 + 3\ 957\ 028.6 - 4\ 649\ 028.6)/46$$

$$R_{20x} = 73\ 800.0$$

Calculation of Truss-Member Forces

1)



$$\Sigma F_x = 0 = 72800 - F_2 - F_1 \cos 25.6$$

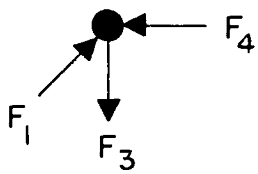
$$\Sigma F_y = 0 = 10195.2 - F_1 \sin 25.6$$

$$F_1 = 23\ 593.5$$

$$F_2 = 51\ 522.9$$

---

2)



$$\Sigma F_x = 0 = F_1 \cos 25.6 - F_4$$

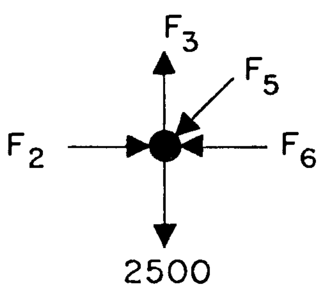
$$\Sigma F_y = 0 = F_1 \sin 25.6 - F_3$$

$$F_3 = 10\ 195.2$$

$$F_4 = 21\ 277.0$$

---

3)



$$\Sigma F_x = 0 = F_2 - F_6 - F_5 \cos 25.6$$

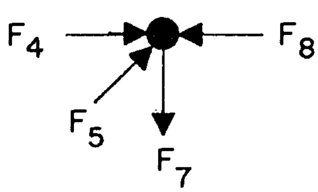
$$\Sigma F_y = 0 = F_3 - 2500 - F_5 \sin 25.6$$

$$F_5 = 17\ 808.1$$

$$F_6 = 35\ 463.4$$

---

4)



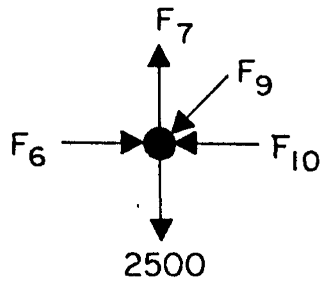
$$\Sigma F_x = 0 = F_4 + F_5 \cos 25.6 - F_8$$

$$\Sigma F_y = 0 = F_5 \sin 25.6 - F_7$$

$$F_7 = 7695.2$$

$$F_8 = 37\ 336.6$$

5)



$$\Sigma F_x = 0 = F_6 - F_{10} - F_9 \cos 25.6$$

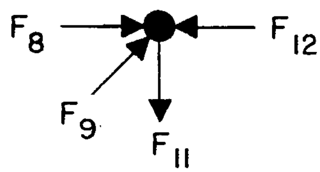
$$\Sigma F_y = 0 = F_7 - 2500 - F_9 \sin 25.6$$

$$F_9 = 12\,022.7$$

$$F_{10} = 24\,621.1$$

---

6)



$$\Sigma F_x = 0 = F_8 + F_9 \cos 25.6 - F_{12}$$

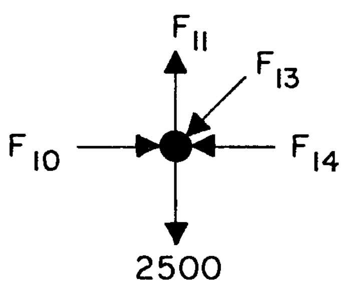
$$\Sigma F_y = 0 = F_9 \sin 25.6 - F_{11}$$

$$F_{11} = 5195.2$$

$$F_{12} = 48\,178.9$$

---

7)



$$\Sigma F_x = 0 = F_{10} - F_{13} \cos 25.6 - F_{14}$$

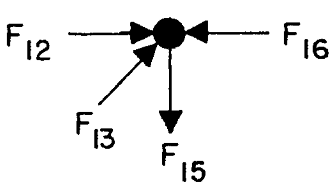
$$\Sigma F_y = 0 = F_{11} - F_{13} \sin 25.6 - 2500$$

$$F_{13} = 6237.2$$

$$F_{14} = 18\,996.3$$

---

8)



$$\Sigma F_x = 0 = F_{12} + F_{13} \cos 25.6 - F_{16}$$

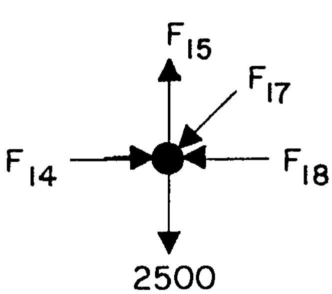
$$\Sigma F_y = 0 = F_{13} \sin 25.6 - F_{15}$$

$$F_{15} = 2695.2$$

$$F_{16} = 53\,803.7$$

---

9)



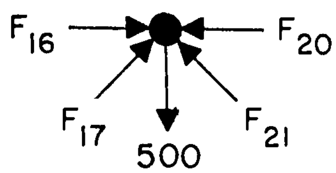
$$\Sigma F_x = 0 = F_{14} - F_{17} \cos 51.9 - F_{18}$$

$$\Sigma F_y = 0 = F_{15} - F_{17} \sin 51.9 - 2500$$

$$F_{17} = 247.9$$

$$F_{18} = 18\,772.7$$

10)



$$\Sigma F_x = 0 = F_{16} + F_{17} \cos 51.9 - F_{20} - F_{21} \cos 51.9$$

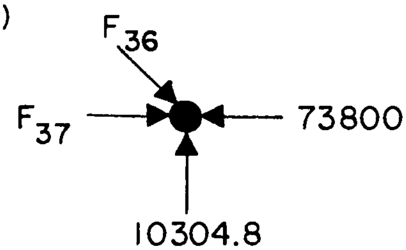
$$\Sigma F_y = 0 = F_{17} \sin 51.9 + F_{21} \sin 51.9 - 500$$

$$F_{20} = 53\ 718.0$$

$$F_{21} = 386.9$$

---

20)



$$\Sigma F_x = 0 = F_{37} + F_{36} \cos 25.6 - 73\ 800$$

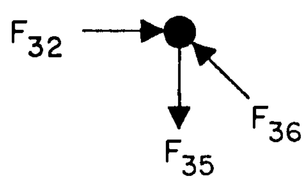
$$\Sigma F_y = 0 = 10\ 304.8 - F_{36} \sin 25.6$$

$$F_{36} = 23\ 846.9$$

$$F_{37} = 52\ 294.4$$

---

18)



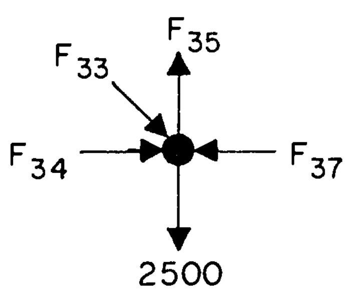
$$\Sigma F_x = 0 = F_{32} - F_{36} \cos 25.6$$

$$\Sigma F_y = 0 = F_{36} \sin 25.6 - F_{35}$$

$$F_{35} = 10\ 304.8$$

$$F_{32} = 21\ 505.6$$

I9)



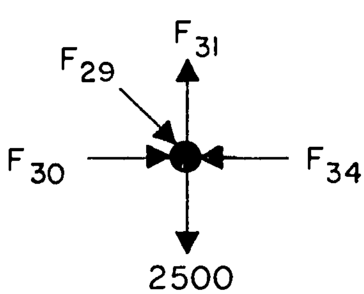
$$\Sigma F_x = 0 = F_{34} + F_{33} \cos 25.6 - F_{37}$$

$$\Sigma F_y = 0 = F_{35} - F_{33} \sin 25.6 - 2500$$

$$F_{33} = 18\ 061.6$$

$$F_{34} = 36\ 006.2$$

I7)



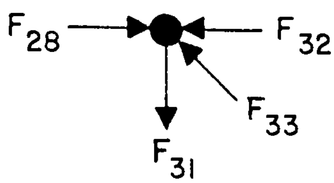
$$\Sigma F_x = 0 = F_{30} + F_{29} \cos 25.6 - F_{34}$$

$$\Sigma F_y = 0 = F_{31} - F_{29} \sin 25.6 - 2500$$

$$F_{29} = 12\ 276.1$$

$$F_{30} = 27\ 935.4$$

16)



$$\Sigma F_x = 0 = F_{28} - F_{32} - F_{33} \cos 25.6$$

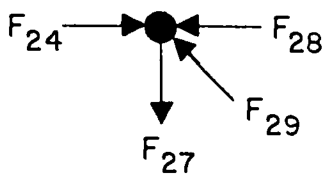
$$\Sigma F_y = 0 = F_{33} \sin 25.6 - F_{31}$$

$$F_{31} = 7804.8$$

$$F_{28} = 37\ 793.8$$

---

14)



$$\Sigma F_x = 0 = F_{24} - F_{28} - F_{29} \cos 25.6$$

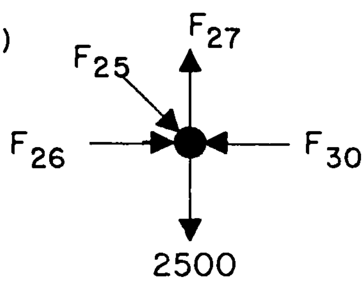
$$\Sigma F_y = 0 = F_{29} \sin 25.6 - F_{27}$$

$$F_{27} = 5304.8$$

$$F_{24} = 48\ 864.6$$

---

15)



$$\Sigma F_x = 0 = F_{26} + F_{25} \cos 25.6 - F_{30}$$

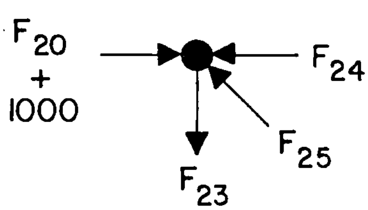
$$\Sigma F_y = 0 = F_{27} - F_{25} \sin 25.6 - 2500$$

$$F_{26} = 19\ 081.9$$

$$F_{25} = 6490.7$$

---

12)



$$\Sigma F_x = 0 = F_{20} + 1000 - F_{24} - F_{25} \cos 25.6$$

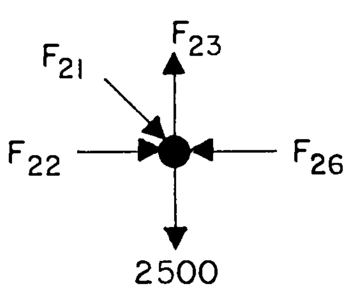
$$\Sigma F_y = 0 = F_{25} \sin 25.6 - F_{23}$$

$$F_{23} = 2804.8$$

$$F_{20} = 53\ 718.0$$

---

13)



$$\Sigma F_x = 0 = F_{22} + F_{21} \cos 51.9 - F_{26}$$

$$\Sigma F_y = 0 = F_{23} - F_{21} \sin 51.9 - 2500$$

$$F_{21} = 386.9$$

$$F_{22} = 18\ 732.9$$

TABLE 4

Truss-Member		Calculated Force	Force-Direction
Lower	2	- 51 522.9	Compression
	6	- 35 463.4	
	10	- 24 621.1	
	14	- 18 996.3	
	18	- 18 772.7	
	22	- 18 732.9	
	26	- 19 081.9	
	30	- 24 935.4	
	34	- 36 006.2	
	37	- 52 294.4	
Upper	4	- 21 277.0	
	8	- 37 336.6	
	12	- 48 178.9	
	16	- 53 803.7	
	20	- 53 718.0	
	24	- 48 864.6	
	28	- 37 793.8	
	32	- 21 505.6	
Diagonal	1	- 23 593.5	
	5	- 17 808.1	
	9	- 12 022.7	
	13	- 6 237.2	
	17	- 247.9	
	21	- 386.9	
	25	- 6 490.7	
	29	- 12 276.1	
	33	- 18 061.6	
	36	- 23 846.9	

TABLE 4 (Cont'd)

Truss-Member	Calculated Force	Force-Direction
Vertical 3	10 195.2	Tension
7	7 695.2	
11	5 195.2	
15	2 695.2	
19	500.0	
23	2 804.8	
27	5 304.8	
31	7 804.8	
35	10 304.8	

These calculations show that compressive forces are maintained in all lower, upper and diagonal members. All vertical members also retain their tensile forces. The degree of change, compared to forces calculated under part B., is minimal and the margin of safety has only slightly decreased and is now at 6200 lbs. versus 6400 lbs. This represents a 3% change and can be ignored.

D. Computation of Forces in Truss-Members with Spring-Force (to be calculated), Internal Force of 1000 lbs. and Internal Pressure of 150 psig.

1) Calculate spring-force

2) Draw diagram of idealized truss (Fig. 16)

3) Calculate by method of joints, forces in individual members

4) Check on calculations.

<u>Calculation of Spring-Force</u>

Spring Constant: K_S = 1800 lb./in.

Tank Expansion Coefficient: K_T = .0058 in./10 psig.

Total Expansion of Tank at 150 psig:

$$\Delta L = (.0058)(15) = .087 \text{ in.}$$

Change in Spring-Force due to .087 in Expansion:

$$\Delta F = (1800)(.087) = 156.6 \text{ lbs.}$$

New Spring-Force at 150 psig.:

$$F_2 = 72\,800 - 156.6 = 72\,643.4 \text{ lbs.}$$

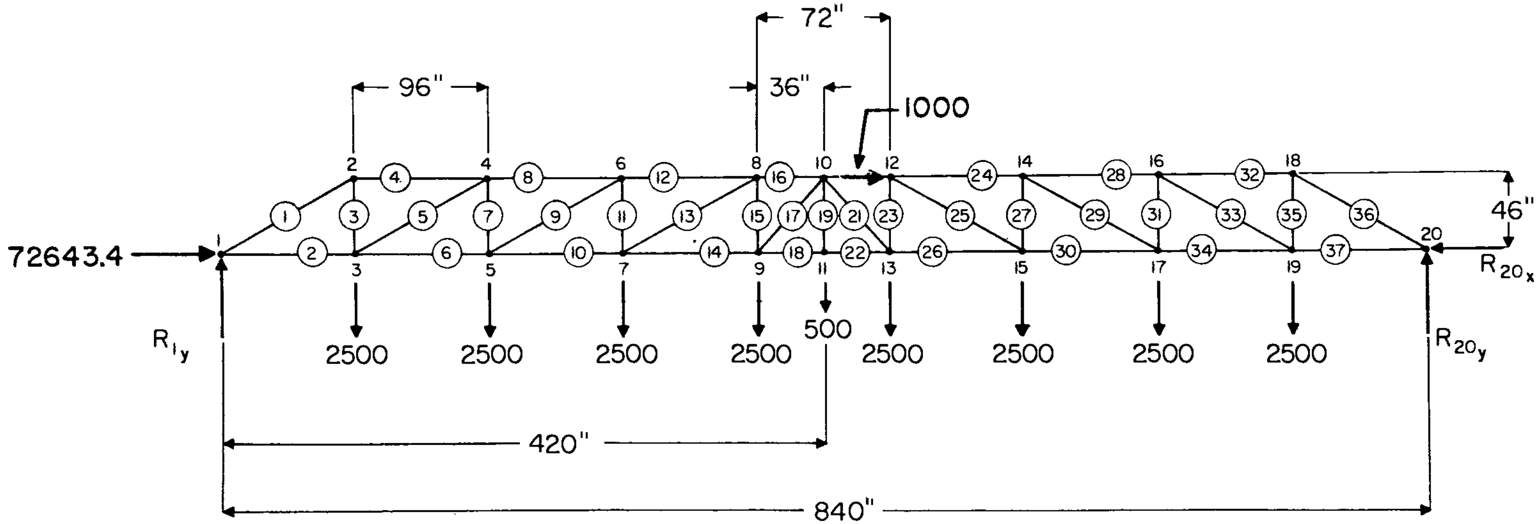
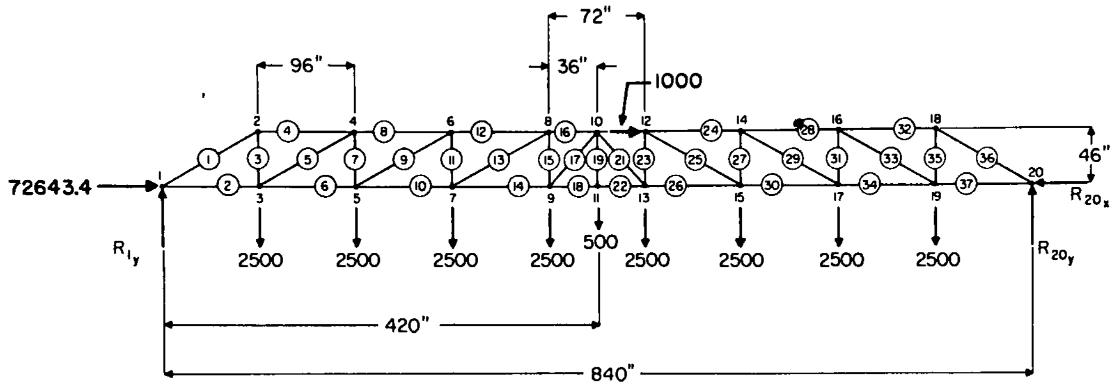


Fig. 16

Calculation of Reaction Forces



$$\Sigma M_{20} = 0 = R_{1y}\ (840)+(1000)(46)-(20500)(420)$$

$$Rly = 10195.2$$

$$\Sigma M_{1} = 0 = (20500)(420)+(1000)(46)- R_{20y}(840)$$

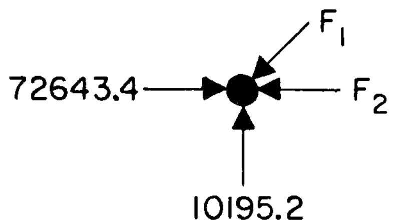
$$R_{20y} = 10\ 304.8$$

$$\Sigma M_{12} = 0 = R_{1y}(456)+ R_{20x}(46)-(72643.4)(46)-(20500)(36) - R_{20y}(384)$$

$$R_{20x} = 73\ 643.4$$

Calculation of Truss-Member Forces

1)



$$\Sigma F_x = 0 = 72643.4 - F_2 - F_1 \cos 25.6$$

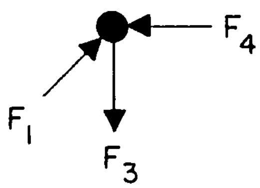
$$\Sigma F_y = 0 = 10195.2 - F_1 \sin 25.6$$

$$F_1 = 23593.5$$

$$F_2 = 51366.4$$

---

2)



$$\Sigma F_x = 0 = F_1 \cos 25.6 - F_4$$

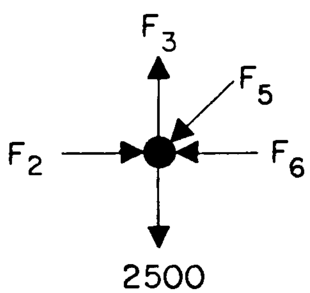
$$\Sigma F_y = 0 = F_1 \sin 25.6 - F_3$$

$$F_3 = 10195.2$$

$$F_4 = 21277.0$$

---

3)



$$\Sigma F_x = 0 = F_2 - F_6 - F_5 \cos 25.6$$

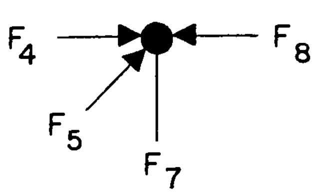
$$\Sigma F_y = 0 = F_3 - 2500 - F_5 \sin 25.6$$

$$F_5 = 17808.1$$

$$F_6 = 35306.8$$

---

4)



$$\Sigma F_x = 0 = F_4 + F_5 \cos 25.6 - F_8$$

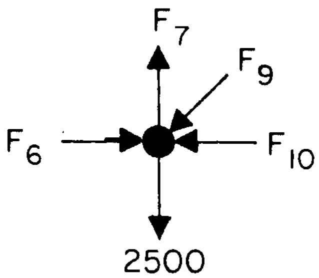
$$\Sigma F_y = 0 = F_5 \sin 25.6 - F_7$$

$$F_7 = 7695.2$$

$$F_8 = 37336.6$$

---

5)



$$\Sigma F_x = 0 = F_6 - F_{10} - F_9 \cos 25.6$$

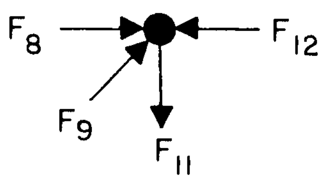
$$\Sigma F_y = 0 = F_7 - 2500 - F_9 \sin 25.6$$

$$F_9 = 12022.7$$

$$F_{10} = 24464.5$$

---

6)



$$\Sigma F_x = 0 = F_8 + F_9 \cos 25.6 - F_{12}$$

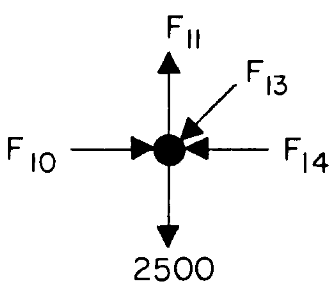
$$\Sigma F_y = 0 = F_9 \sin 25.6 - F_{11}$$

$$F_{11} = 5195.2$$

$$F_{12} = 48178.9$$

---

7)



$$\Sigma F_x = 0 = F_{10} - F_{13} \cos 25.6 - F_{14}$$

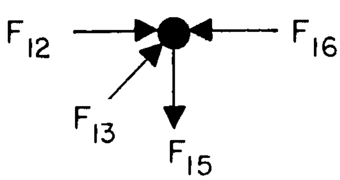
$$\Sigma F_y = 0 = F_{11} - F_{13} \sin 25.6 - 2500$$

$$F_{13} = 6237.2$$

$$F_{14} = 18839.7$$

---

8)



$$\Sigma F_x = 0 = F_{12} + F_{13} \cos 25.6 - F_{16}$$

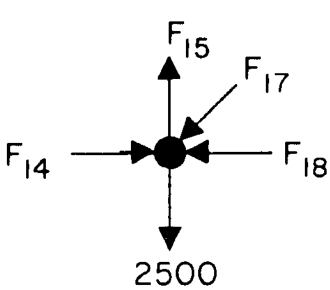
$$\Sigma F_y = 0 = F_{13} \sin 25.6 - F_{15}$$

$$F_{15} = 2695.2$$

$$F_{16} = 53803.7$$

---

9)



$$\Sigma F_x = 0 = F_{14} - F_{17} \cos 51.9 - F_{18}$$

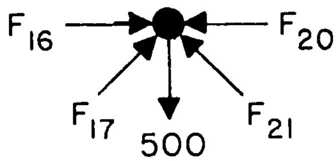
$$\Sigma F_y = 0 = F_{15} - F_{17} \sin 51.9 - 2500$$

$$F_{17} = 247.9$$

$$F_{18} = 18686.9$$

---

10)



$$\Sigma F_x = 0 = F_{16} + F_{17}\cos 51.9 - F_{20} - F_{21}\cos 51.9$$

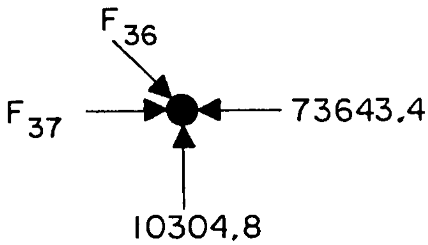
$$\Sigma F_y = 0 = F_{17}\sin 51.9 + F_{21}\sin 51.9 - 500$$

$$F_{20} = 53718.0$$

$$F_{21} = 386.9$$

---

20)



$$\Sigma F_x = 0 = F_{37} + F_{36}\cos 25.6 - 73643.4$$

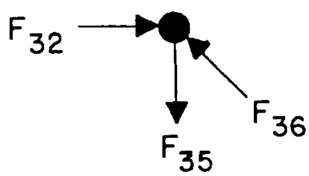
$$\Sigma F_y = 0 = 10304.8 - F_{36}\sin 25.6$$

$$F_{36} = 23846.9$$

$$F_{37} = 52137.8$$

---

18)



$$\Sigma F_x = 0 = F_{32} - F_{36}\cos 25.6$$

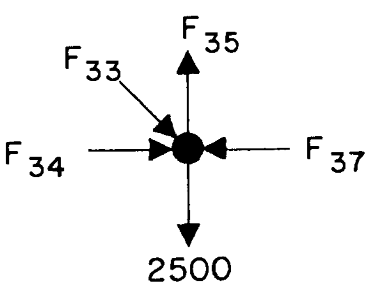
$$\Sigma F_y = 0 = F_{36}\sin 25.6 - F_{35}$$

$$F_{35} = 10304.8$$

$$F_{32} = 21505.6$$

---

19)



$$\Sigma F_x = 0 = F_{34} + F_{33}\cos 25.6 - F_{37}$$

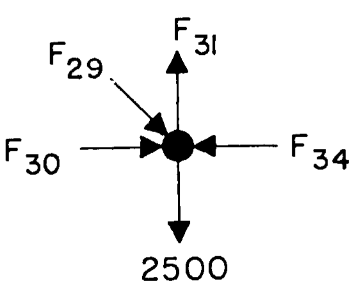
$$\Sigma F_y = 0 = F_{35} - F_{33}\sin 25.6 - 2500$$

$$F_{33} = 18061.6$$

$$F_{34} = 35849.6$$

---

17)



$$\Sigma F_x = 0 = F_{30} + F_{29}\cos 25.6 - F_{34}$$

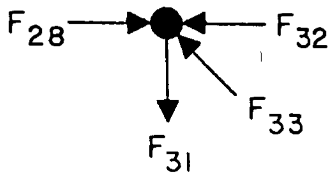
$$\Sigma F_y = 0 = F_{31} - F_{29}\sin 25.6 - 2500$$

$$F_{29} = 12276.1$$

$$F_{30} = 24778.8$$

---

16)



$$\Sigma F_x = 0 = F_{28} - F_{32} - F_{33}\cos 25.6$$

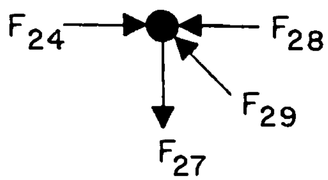
$$\Sigma F_y = 0 = F_{33}\sin 25.6 - F_{31}$$

$$F_{31} = 7804.8$$

$$F_{28} = 37793.8$$

---

14)



$$\Sigma F_x = 0 = F_{24} - F_{28} - F_{29}\cos 25.6$$

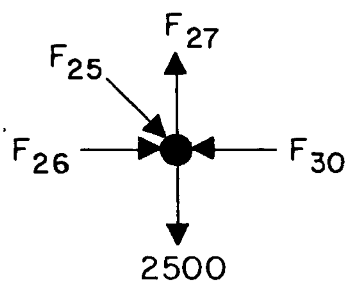
$$\Sigma F_y = 0 = F_{29}\sin 25.6 - F_{27}$$

$$F_{27} = 5304.8$$

$$F_{24} = 48864.6$$

---

15)



$$\Sigma F_x = 0 = F_{26} + F_{25}\cos 25.6 - F_{30}$$

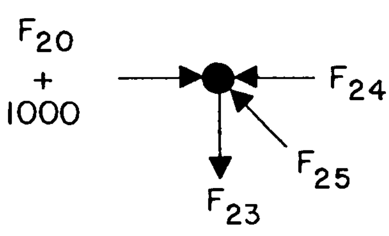
$$\Sigma F_y = 0 = F_{27} - F_{25}\sin 25.6 - 2500$$

$$F_{26} = 18925.4$$

$$F_{25} = 6490.7$$

---

12)



$$\Sigma F_x = 0 = F_{20} + 1000 - F_{24} - F_{25}\cos 25.6$$

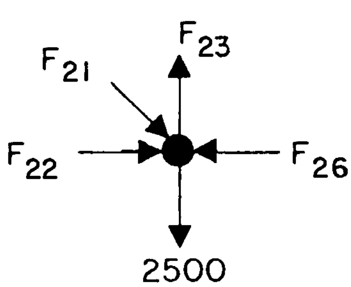
$$\Sigma F_y = 0 = F_{25}\sin 25.6 - F_{23}$$

$$F_{23} = 2804.8$$

$$F_{20} = 53718.0$$

---

13)



$$\Sigma F_x = 0 = F_{22} + F_{21}\cos 51.9 - F_{26}$$

$$\Sigma F_y = 0 = F_{23} - F_{21}\sin 51.9 - 2500$$

$$F_{21} = 386.9$$

$$F_{22} = 18686.9$$

TABLE 5

Truss-Member		Calculated Force	Force-Direction
Lower	2	- 51 366.4	Compression
	6	- 35 306.7	
	10	- 24 464.5	
	14	- 18 839.7	
	18	- 18 686.9	
	22	- 18 686.9	
	26	- 18 925.4	
	30	- 24 778.8	
	34	- 35 849.6	
	37	- 52 137.8	
Upper	4	- 21 277.0	
	8	- 37 336.6	
	12	- 48 178.9	
	16	- 53 803.7	
	20	- 53 718.0	
	24	- 48 864.6	
	28	- 37 793.8	
	32	- 21 505.6	
Diagonal	1	- 23 593.5	
	5	- 17 808.1	
	9	- 12 022.7	
	13	- 6 237.2	
	17	- 247.9	
	21	- 386.9	
	25	- 6 490.7	
	29	- 12 276.1	
	33	- 18 061.6	
	36	- 23 846.9	

TABLE 5 (Cont'd)

Truss-Member	Calculated Force	Force-Action
Vertical 3	10 195.2	Tension
7	7 695.2	
11	5 195.2	
15	2 695.2	
19	500.0	
23	2 804.8	
27	5 304.8	
31	7 804.8	
35	10 304.8	

These calculations show that only minor changes in forces took place during pressurization of tank. Adequate compression is maintained during this cycle and the safety margin remains high at 6200 lbs.

VII. COMPUTATION OF DISPLACEMENT AT MIDPOINT OF TRUSS

A. With Spring-Force of 72800 lbs. and Internal Force of 1000 lbs.

For the calculations of this displacement, we proceed as follows:

1) Draw a diagram of idealized truss with virtual load at midpoint in the direction of displacement and with reaction forces (Fig. 16).
2) Calculate reaction forces.
3) Calculate virtual forces of all truss members.
4) Calculate: $\Delta = \Sigma \frac{F\mu L}{AE}$

Length, cross sectional area and modulus of elasticity of individual truss members are taken from Table 25. Forces from actual loads are taken from Table 4. Virtual forces μ are calculated in this section.

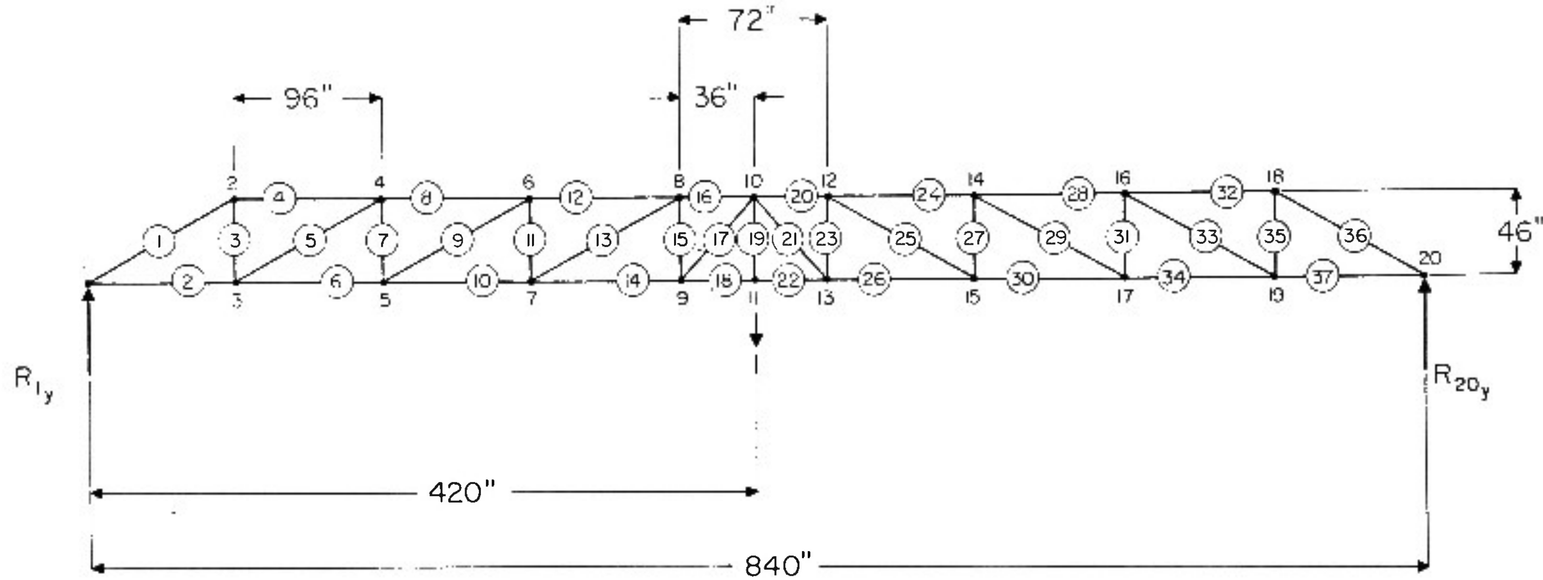
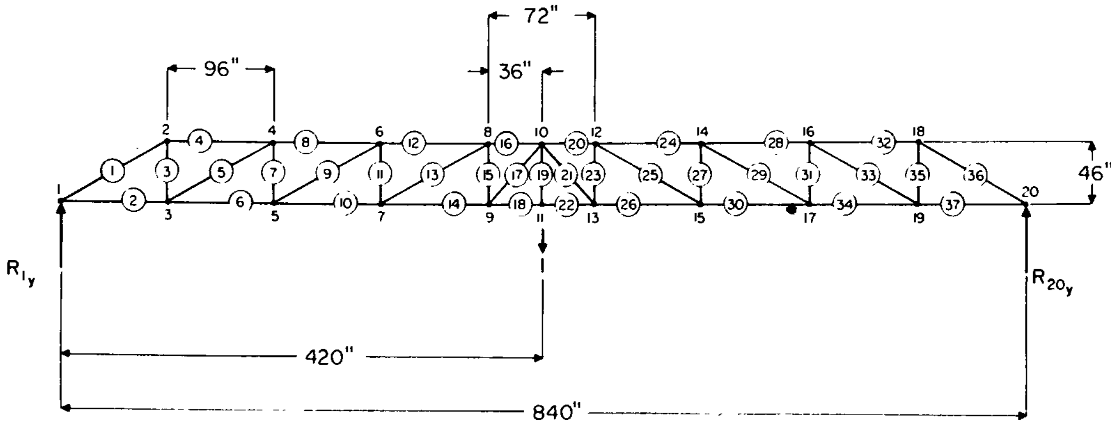


Fig.I

Computation of Reaction Forces



$$\Sigma F_x = 0$$

$$\Sigma F_y = R_{1y} + R_{20y} = 1$$

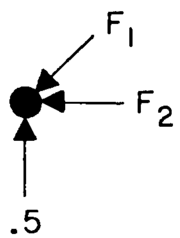
$$\Sigma M_1 = (1)(420) - R_{20y}(840) = 0$$

$$R_{20y} = .5$$

$$R_{1y} = .5$$

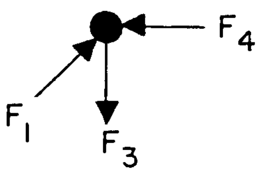
Computation of Truss-Member Forces

1)



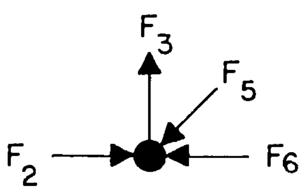
$$\Sigma F_x = 0 = -F_2 - F_1 \cos 25.6$$
$$\Sigma F_y = 0 = .5 - F_1 \sin 25.6$$
$$F_1 = 1.157$$
$$F_2 = -1.04340$$

2)



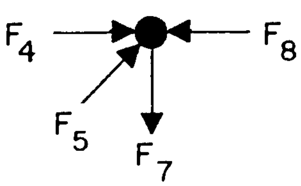
$$\Sigma F_x = 0 = F_1 \cos 25.6 - F_4$$
$$\Sigma F_y = 0 = F_1 \sin 25.6 - F_3$$
$$F_3 = .5$$
$$F_4 = 1.04340$$

3)



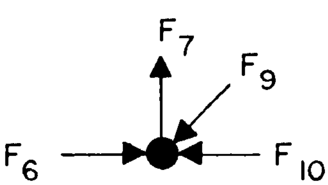
$$\Sigma F_x = 0 = F_2 - F_6 - F_5 \cos 25.6$$
$$\Sigma F_y = 0 = F_3 - F_5 \sin 25.6$$
$$F_5 = 1.157$$
$$F_6 = -2.08687$$

4)



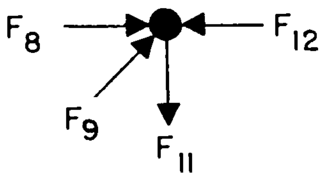
$$\Sigma F_x = 0 = F_4 + F_5 \cos 25.6 - F_8$$
$$\Sigma F_y = 0 = F_5 \sin 25.6 - F_7$$
$$F_7 = .5$$
$$F_8 = 2.08687$$

5)



$$\Sigma F_x = 0 = F_6 - F_{10} - F_9 \cos 25.6$$
$$\Sigma F_y = 0 = F_7 - F_9 \sin 25.6$$
$$F_9 = 1.157$$
$$F_{10} = -3.13035$$

6)



$$\Sigma F_x = 0 = F_8 + F_9 \cos 25.6 - F_{12}$$

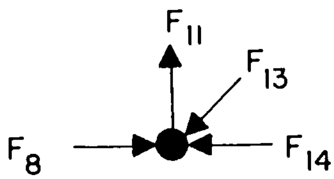
$$\Sigma F_y = 0 = F_9 \sin 25.6 - F_{II}$$

$$F_{11} = .5$$

$$F_{12} = 3.1$$

---

7)



$$\Sigma F_x = 0 = F_{10} - F_{13} \cos 25.6 - F_{14}$$

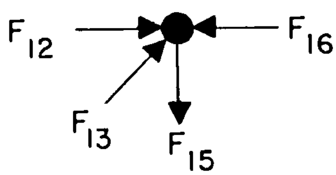
$$\Sigma F_y = 0 = F_{11} - F_{13} \sin 25.6$$

$$F_{13} = 1.157$$

$$F_{14} = -4.2$$

---

8)



$$\Sigma F_x = 0 = F_{12} + F_{13} \cos 25.6 - F_{16}$$

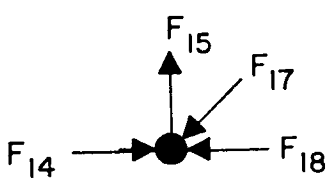
$$\Sigma F_y = 0 = F_{13} \sin 25.6 - F_{15}$$

$$F_{15} = .5$$

$$F_{16} = 4.2$$

---

9)



$$\Sigma F_x = 0 = F_{14} - F_{17} \cos 51.9 - F_{18}$$

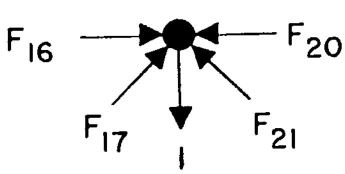
$$\Sigma F_y = 0 = F_{15} - F_{17} \sin 51.9$$

$$F_{17} = .6$$

$$F_{18} = -4.6$$

---

10)



$$\Sigma F_x = 0 = F_{16} + F_{17} \cos 51.9 - F_{20} - F_{21} \cos 51.9$$

$$\Sigma F_y = 0 = F_{17} \sin 51.9 + E_{21} \sin 51.9 - 1$$

$$F_{20} = 4.2$$

$$F_{21} = .6$$

TABLE 6

Truss Member	Calc. Force F	Virtual Force μ	L/AE	FμL/AE
1	- 23 593.5	- 1.157	5.714 -7	1.6144 -2
2	- 51 522.9	+ 1.0434	3.0 -7	- 1.6127 -2
3	+ 10 195.2	+ .5	1.704 -7	8.686 -4
4	- 21 277.0	- 1.0434	4.0 -7	8.880 -3
5	- 17 808.1	- 1.157	5.914 -7	1.2185 -2
6	- 35 463.4	+ 2.08687	4.0 -7	- 2.9603 -2
7	+ 7 695.2	+ .5	1.704 -7	6.556 -4
8	- 37 336.6	- 2.08687	3.0 -7	2.3375 -2
9	- 12 022.7	- 1.157	5.914 -7	8.2265 -3
10	- 24 621.1	+ 3.13035	4.0 -7	- 3.0829 -2
11	+ 5 195.2	+ .5	1.704 -7	4.4263 -4
12	- 48 178.9	- 3.13035	3.0 -7	4.5245 -2
13	- 6 237.2	- 1.157	8.871 -7	6.4017 -3
14	- 18 996.3	+ 4.17383	4.0 -7	- 3.17149 -2
15	+ 2 695.2	+ .5	1.709 -7	2.2963 -4
16	- 53 803.7	- 4.17375	1.0 -7	2.2456 -2
17	- 247.9	- .63492	1.298 -7	2.043 -5
18	- 18 772.7	+ 4.56513	1.0 -7	- 8.5699 -3
19	+ 500.0	+ 1	1.022 -7	5.11 -5
20	- 53 718.0	- 4.17375	1.0 -7	2.242 -2
21	- 386.9	- .63491	1.298 -7	3.1835 -5
22	- 18 732.9	+ 4.56513	1.0 -7	- 8.5518 -3
23	+ 2 804.8	+ .5	1.704 -7	2.3897 -4
24	- 48 864.6	- 3.13035	3.0 -7	4.5889 -2
25	- 6 490.7	- 1.157	8.871 -7	6.6619 -3
26	- 19 081.9	+ 4.17383	4.0 -7	- 3.1858 -2
27	+ 5 304.8	+ .5	1.704 -7	4.5197 -4
28	- 37 793.3	- 2.08687	3.0 -7	2.3661 -2

TABLE 6 (Cont'd)

Truss Member	Calc. Force F	Virtual Force μ	$\frac{L}{AE}$	$\frac{F\mu L}{AE}$
29	- 12 276.1	- 1.157	5.9 -7	8.3999 -3
30	- 24 935.4	+ 3.13035	4.0 -7	- 3.1223 -2
31	+ 7 804.8	+ .5	1.704 -7	6.6496 -4
32	- 21 505.6	- 1.0434	4.0 -7	8.9756 -3
33	- 18 061.6	- 1.157	5.914 -7	1.2358 -2
34	- 36 006.2	+ 2.08687	4.0 -7	- 3.0056 -2
35	+ 10 304.8	+ .5	1.704 -7	8.7848 -4
36	- 23 846.9	- 1.157	5.914 -7	1.6317 -2
37	- 52 294.4	+ 1.0434	3.0 -7	1.6369 -2

Σ .0899

Δ = .0899" Down

These calculations show that the maximum displacement is 0.0899".

During alignment of instruments which are mounted along the truss this displacement must be taken into account.

B. Computation of Displacement at Midpoint of Truss with Spring Force of 72800 lbs, Internal Force of 1000 lbs, and Internal Pressure of 150 psig.

For the calculations we proceed as in part A.

1) Draw diagram of idealized truss with virtual load at midpoint in the direction of displacement and with reaction forces (Fig. 16).

2) Calculate reaction forces (page 53).

3) Calculate virtual forces of all truss members (Table 6).

4) Calculate: $\Delta = \Sigma \frac{F\mu L}{AE}$

Length, cross sectional area and modulus of elasticity of individual members is taken from Table 25. Forces from actual loads are taken from Table 5. Virtual forces μ are taken from Table 6.

TABLE 7

Truss Member	Calc. Force F	Virtual Force μ	$\frac{L}{AE}$		$\frac{F\mu L}{AE}$	
1	- 23 593.5	- 1.157	5.914	-7	1.6144	-2
2	- 51 366.4	+ 1.0434	3.0	-7	- 1.6078	-2
3	+ 10 195.2	+ .5	1.704	-7	8.686	-4
4	- 21 277.0	- 1.0434	4.0	-7	8.880	-3
5	- 17 808.1	- 1.157	5.914	-7	1.2185	-2
6	- 35 306.7	+ 2.08687	4.0	-7	-2.9472	-2
7	+ 7 695.2	+ .5	1.704	-7	6.556	-4
8	- 37 336.6	- 2.08687	3.0	-7	2.3375	-2
9	- 12 022.7	- 1.157	5.914	-7	- 8.2265	-3
10	- 24 464.5	+ 3.13035	4.0	-7	- 3.0629	-2
11	+ 5195.2	+ .5	1.704	-7	4.4263	-4
12	- 48 178.9	- 3.13035	3.0	-7	4.5245	-2
13	- 6 237.2	- 1.157	8.871	-7	- 6.4017	-2
14	- 18 839.7	+ 4.173	4.0	-7	- 3.1453	-2
15	+ 2 695.2	+ .5	1.709	-7	2.2963	-4
16	- 53 803.7	- 4.17375	1.0	-7	2.2456	-2
17	- 247.9	- .63492	1.298	-7	2.043	-5
18	- 18 686.9	+ 4.56513	1.0	-7	- 8.5308	-3
19	+ 500.0	+ 1.0	1.022	-7	5.11	-5
20	- 53 718.0	- 4.17375	1.0	-7	2.242	-2
21	- 386.9	- .63492	1.298	-7	3.1885	-5
22	- 18 686.9	+ 4.56513	1.0	-7	- 8.5308	-3
23	+ 2 804.82	+ .5	1.704	-7	2.3897	-4
24	- 48 864.6	- 3.13035	3.0	-7	4.5887	-2
25	- 6 490.7	- 1.157	8.871	-7	6.6619	-3
26	- 18 925.4	+ 4.17383	4.0	-7	- 3.15958	-2
27	+ 5 304.8	+ .5	1.704	-7	4.5197	-4
28	- 37 793.8	- 2.08687	3.0	-7	2.3661	-2

TABLE 7 (Cont'd)

Truss Member	Calc. Force F	Virtual Force S	L / AE		FSL / AE	
29	- 12 276.1	- 1.157	5.914	-7	8.3999	-3
30	- 24 778.8	+ 3.13035	4.0	-7	- 3.1026	-2
31	7 804.8	+ .5	1.704	-7	6.649	-4
32	- 21 505.6	- 1.0434	4.0	-7	8.9756	-3
33	- 18 061.6	- 1.157	5.914	-7	1.2358	-2
34	- 35 849.6	+ 2.08687	4.0	-7	2.9925	-2
35	10 304.8	+ .5	1.704	-7	8.7848	-4
36	23 846.9	- 1.157	5.914	-7	1.6317	-2
37	- 52 137.8	+ 1.0434	3.0	-7	1.6320	-2

Σ .091

Δ = .091" Down

These calculations show that maximum displacement under pressurization of vessel is only .002" more than without pressure. This is, on the face of it, surprising. One would expect more. It agrees, however, exactly with measurements taken on the actual structure.

VIII. COMPUTATION OF FORCES AND DISPLACEMENTS IN TRUSS-MEMBERS USING THE SAP IV ANALYSIS PROGRAM

A. Gravity loads of Truss Members Included in Loads at nodes

a) Without Compressive Spring Force

TABLE 8

Truss Member		SAP IV	Force Direction
Lower	2,37	21391.3	Tension
	6,34	37 565.2	
	10,30	48 521.7	
	14,26	54 260.9	
	18,22	54 456.5	
Upper	4,32	- 21 391.3	Compression
	8,28	- 37 565.2	
	12,24	- 48 521.7	
	16,20	- 54 260.9	
Diag.	1,36	- 23 720.3	
	5,33	- 17 934.8	
	9,29	- 12 149.4	
	13,25	- 6 363.9	
	17,21	- 317.5	
Vert.	3,35	10 250.0	Tension
	7,31	7 750.0	
	11,27	5 250.0	
	15,23	2 750.0	
	19	500.0	

b) Computation of Forces in Truss Members with Spring Force of 72000 lbs.

TABLE 9

Truss-Member		SAP IV	• Force-Direction
Lower	2,37	- 51 408.7	Compression
	6,34	- 35 234.8	
	10,30	- 24 278.3	
	14,26	- 18 539.1	
	18,22	- 18 343.5	
Upper	4,32	- 21 391.3	
	8,28	- 37 565.2	
	12,24	- 48 521.7	
	16,20	- 54 260.9	
Diagonal	1,36	- 23 720.3	
	5,33	- 17 934.8	
	9,29	- 12 149.4	
	13,25	- 6 363.9	
	17,21	- 317.5	
Vertical	3,35	10 250.0	Tension
	7,31	7 750.0	
	11,27	5 250.0	
	15,23	2 750.0	
	19	500.0	

c) Computation of Forces in Truss-Members with Spring Force of 72800 lbs. and Internal Force of 1000 lbs.

TABLE 10

Truss-Member		SAP IV	Force-Direction
Lower	2	- 51 522.9	Compression
	6	- 35 463.4	
	10	- 24 621.1	
	14	- 18 996.3	
	18	- 18 843.5	
	22	- 18 843.5	
	26	- 19 081.9	
	30	- 24 935.4	
	34	- 36 006.2	
	37	- 52 294.4	
Upper	4	- 21 277.0	
	8	- 37 336.7	
	12	- 48 178.9	
	16	- 53 803.7	
	20	- 53 718.0	
	24	- 48 864.6	
	28	- 37 793.8	
	32	- 21 505.6	
Diagonal	1	- 23 593.5	
	5	- 17 808.1	
	9	- 12 022.7	
	13	- 6 237.2	
	17	- 247.9	
	21	- 386.9	
	25	- 6 490.7	
	29	- 12 276.1	

TABLE 10 (Cont'd)

Truss-Member		SAP IV	Force-Direction
Diagonal	33	- 18 061.6	Compression
	36	- 23 846.9	
Vertical	3	10 195.2	Tension
	7	7 695.2	
	11	5 195.2	
	15	2 695.2	
	19	500.0	
	23	2 804.8	
	27	5 305.8	
	31	7 804.8	
	35	10 304.8	

Displacement at Midpoint of Span, Node 11:

.057" Down

d) Computation of Forces in Truss-Members with Spring Force of 72800 lbs., 1000 lbs. Internal Force and Internal Pressure of 150 psig.

TABLE 11

Truss-Member		SAP IV	Force-Direction
Lower	2	- 51 366.4	Compression
	6	- 35 306.8	
	10	- 24 464.5	
	14	- 18 839.7	
	18	- 18 686.9	
	22	- 18 686.9	
	26	- 18 925.4	
	30	- 24 778.8	
	34	- 35 849.6	
	37	- 52 137.8	
Upper	4	- 21 277.0	
	8	- 37 336.7	
	12	- 48 178.9	
	16	- 53 803.7	
	20	- 53 718.0	
	24	- 48 864.6	
	28	- 37 793.8	
	32	- 21 505.6	
Diagonal	1	- 23 593.5	
	5	- 17 808.1	
	9	- 12 022.7	
	13	- 6 237.2	
	17	- 247.9	
	21	- 386.9	
	25	- 6 490.7	

TABLE 11 (Cont'd)

Truss-Member		SAP IV	Force-Direction
Diagonal	29	- 12 276.1	Compression
	33	- 18 061.6	↓
	36	- 23 846.9	↓
Vertical	3	10 195.2	Tension
	7	7 695.2	↓
	11	5 195.2	↓
	15	2 695.2	↓
	19	500.0	↓
	23	2 804.8	↓
	21	5 304.8	↓
	31	7 804.8	↓
	35	10 304.8	↓

Displacement at Midpoint of Span, Node 11:

$$\Delta = .058'' \quad \text{Down}$$

B. Gravity Loads of Truss-Members Added by SAP IV and All Other Loads Concentrated at Nodes

a) Without Compressive Spring-Force

TABLE 12

Truss-Member		SAP IV	Force-Direction
Lower	2,37	20 581.9	Tension
	6,34	36 451.6	
	10,30	47 222.2	
	14,26	52 872.0	
	18,22	53 336.8	
Upper	4,32	- 20 581.9	Compression
	8,28	- 36 451.6	
	12,24	- 47 222.2	
	16,20	- 52 872.0	
Diagonal	1,36	- 22 822.7	
	5,33	- 17 597.4	
	9,29	- 11 943.2	
	13,25	- 6 264.9	
	17,21	- 754.1	
Vertical	3,35	9 464.1	Tension
	7,31	6 959.0	
	11,27	4 453.9	
	15,23	2 237.6	
	19	719.9	

b) Computation of Forces in Truss-Member with
Spring Force of 72800 lbs.

TABLE 13

Truss-Member		SAP IV	Force-Direction
Lower	2,37	- 52 218.1	Compression
	6,34	- 36 348.4	
	10,30	- 25 577.8	
	14,26	- 19 927.9	
	18,22	- 19 463.2	
Upper	4,32	- 20 581.9	
	8,28	- 36 451.6	
	12,24	- 47 222.2	
	16,20	- 52 872.0	
Diagonal	1,36	- 22 822.7	
	5,33	- 17 597.4	
	9,29	- 11 943.2	
	13,25	- 6 264.9	
	17,21	- 754.1	
Vertical	3,35	9 464.1	Tension
	7,31	6 959.0	
	11,27	4 453.9	
	15,23	2 237.6	
	19	719.9	

c) Computation of Forces in Truss-Members with Spring Force of 72800 lbs. and Internal Force of 1000 lbs.

TABLE 14

Truss-Member		SAP IV	Force-Direction
Lower	2	- 52 332.4	Compression
	6	- 36 577.0	
	10	- 25 920.7	
	14	- 20 385.1	
	18	- 19 963.2	
	22	- 19 963.2	
	26	- 20 470.9	
	30	- 26 234.9	
	34	- 37 119.9	
	37	- 53 103.8	
Upper	4	- 20 467.6	
	8	- 36 222.9	
	12	- 46 879.3	
	16	- 52 414.9	
	20	- 52 329.2	
	24	- 47 565.0	
	28	- 36 680.1	
	32	- 20 696.2	
Diagonal	1	- 22 696.0	
	5	- 17 470.7	
	9	- 11 816.5	
	13	- 6 138.2	
	17	- 684.6	
	21	- 823.7	
	25	- 6 391.7	

TABLE 14 (Cont'd)

Truss-Member		SAP IV	Force-Direction
Diagonal	29	- 12 069.9	Compression
	33	- 17 724.2	
	36	- 22 949.5	
Vertical	3	9 409.4	Tension
	7	6 904.3	
	11	4 399.2	
	15	2 182.9	
	19	719.9	
	23	2 292.4	
	27	4 508.7	
	31	7 013.8	
	35	9 518.9	

Displacement at Midpoint of Span, Node 11:

Δ = .038" Down

d) Computation of Forces in Truss-Members with Spring-Force of 72000 lbs., 1000 lbs. Internal Force and Internal Pressure of 150 psig.

TABLE 15

Truss-Member		SAP IV	Force-Direction
Lower	2	- 52 175.8	Compression
	6	- 36 420.4	
	10	- 25 764.1	
	14	- 20 228.5	
	18	- 19 806.6	
	22	- 19 806.6	
	26	- 20 314.3	
	30	- 26 078.4	
	34	- 36 963.3	
	37	- 52 947.2	
Upper	4	- 20 467.6	
	8	- 36 222.9	
	12	- 46 879.3	
	16	- 52 414.9	
	20	- 52 329.2	
	24	- 47 565.0	
	28	- 36 680.1	
	32	- 20 696.2	
Diagonal	1	- 22 696.0	
	5	- 17 470.7	
	9	- 11 816.5	
	13	- 6 138.2	
	17	- 684.6	
	21	- 823.6	
	25	- 6 391.7	

TABLE 15 (Cont'd)

Truss-Member		SAP IV	Force-Direction
Diagonal	29	- 12 096.9	Compression
	33	- 17 724.2	
	36	- 22 949.5	
Vertical	3	9 409.4	Tension
	7	6 904.3	
	11	4 399.2	
	15	2 182.9	
	19	719.9	
	23	2 292.4	
	27	4 508.7	
	31	7 013.8	
	35	9 518.9	

Displacement at Midpoint of Span, Node 11:

Δ = .039" Down

IX. COMPARISON OF FORCES AND DISPLACEMENTS OF TRUSS-MEMBERS. EXACT SOLUTIONS VERSUS SAP IV SOLUTIONS.

A. Gravity Loads of Truss-Members Included in Loads at Nodes.

a) Without Compressive Spring-Force

TABLE 16

Truss-Member		SAP IV	EXACT	%-Error
Lower	2,37	21 391.3	21 391.3	0.0
	6,34	37 565.2	37 565.2	0.0
	10,30	48 521.7	48 521.7	0.0
	14,26	54 260.9	54 260.9	0.0
	18,22	54 456.5	54 456.5	0.0
Upper	4,32	- 21 391.3	- 21 391.3	0.0
	8,28	- 37 565.2	- 37 565.2	0.0
	12,24	- 48 521.7	- 48 521.7	0.0
	16,20	- 54 260.9	- 54 260.9	0.0
Diag.	1,36	- 23 720.3	- 23 720.3	0.0
	5,33	- 17 934.8	- 17 934.8	0.0
	9,29	- 12 149.4	- 12 149.4	0.0
	13,25	- 6 363.9	- 6 363.9	0.0
	17,21	- 317.5	- 317.5	0.0
Vert.	3,35	10 250.0	10 250.0	0.0
	7,31	7 750.0	7 750.0	0.0
	11,27	5 250.0	5 250.0	0.0
	15,23	2 750.0	2 750.0	0.0
	19	500.0	500.0	0.0

b) With Compressive Spring-Force of 72800 lbs.

TABLE 17

Truss-Member		SAP IV	EXACT	%-Error
Lower	2,37	- 51 408.7	- 51 408.7	0.0
	6,34	- 35 234.8	- 35 234.8	0.0
	10,30	- 24 278.3	- 24 278.3	0.0
	14,26	- 18 539.1	- 18 539.3	0.0
	18,22	- 18 343.5	- 18 343.6	0.0
Upper	4,32	- 21 391.3	- 21 391.3	0.0
	8,28	- 37 565.2	- 37 565.2	0.0
	12,24	- 48 521.7	- 48 521.7	0.0
	16,20	- 54 260.9	- 54 260.9	0.0
Diag.	1,36	- 23 720.3	- 23 720.3	0.0
	5,33	- 17 934.8	- 17 934.8	0.0
	9,29	- 12 149.4	- 12 149.4	0.0
	13,25	- 6 363.9	- 6 363.9	0.0
	17,21	- 317.5	- 317.5	0.0
Vert.	3,35	10 250.0	10 250.0	0.0
	7,31	7 750.0	7 750.0	0.0
	11,27	5 250.0	5 250.0	0.0
	15,23	2 750.0	2 750.0	0.0
	19	500.0	500.0	0.0

c) With Spring Force of 72800 lbs. and Internal Force of 1000 lbs.

TABLE 18

Truss Member		SAP IV	EXACT	%-Error
Lower	2	- 51 522.9	- 51 522.9	0.0
	6	- 35 463.4	- 35 463.4	0.0
	10	- 24 621.1	- 24 621.1	0.0
	14	- 18 996.3	- 18 996.3	0.0
	18	- 18 843.5	- 18 772.7	0.4
	22	- 18 843.5	- 18 732.9	0.6
	26	- 19 081.9	- 19 081.9	0.0
	30	- 24 935.4	- 24 935.4	0.0
	34	- 36 006.2	- 36 006.2	0.0
	37	- 52 294.4	- 52 294.4	0.0
Upper	4	- 21 277.0	- 21 277.0	0.0
	8	- 37 336.7	- 37 336.7	0.0
	12	- 48 178.9	- 48 178.9	0.0
	16	- 53 803.7	- 53 803.7	0.0
	20	- 53 718.0	- 53 718.0	0.0
	24	- 48 864,6	- 48 864.6	0.0
	28	- 37 793.8	- 37 793.8	0.0
	32	- 21 505.6	- 21 505.6	0.0
Diag.	1	- 23 572.5	- 23 593.5	0.0
	5	- 17 808.1	- 17 808.1	0.0
	9	- 12 022.7	- 12 022.7	0.0
	13	- 6 237.2	- 6 237.2	0.0
	17	- 247.9	- 247.9	0.0
	21	- 386.9	- 386.9	0.0
	25	- 6 490.7	- 6 490.7	0.0
	29	- 12 276.1	- 12 276.1	0.0

TABLE 18 (Cont'd)

Truss-Member		SAP IV	EXACT	%-Error
Diagonal	33	- 18 061.6	- 18 061.6	0.0
	36	- 23 846 9	- 23 846.9	0.0
Vertical	3	10 195.2	10 195.2	0.0
	7	7 695.2	7 695.2	0.0
	11	5 195.2	5 195.2	0.0
	15	2 695.2	2 695.2	0.0
	19	500.0	500.0	0.0
	23	2 804.8	2 804.8	0.0
	27	5 304.8	5 304.8	0.0
	31	7 804.8	7 804.8	0.0
	35	10 304.8	10 304.8	0.0

Displacement of Truss at Node 11:

SAP IV	EXACT	%-Error
.0570"	.0899"	36.6

d) With Spring Force of 72800 lbs., 1000 lbs Internal Force and Internal Pressure of 150 psig.

TABLE 19

Truss-Member		SAP IV	EXACT	%-Error
Lower	2	- 51 366.4	- 51 366.4	0.0
	6	- 35 306.8	- 35 306.7	0.0
	10	- 24 464.5	- 24 464.5	0.0
	14	- 18 839.7	- 18 839.7	0.0
	18	- 18 686.9	- 18 686.9	0.0
	22	- 18 686.9	- 18 686.9	0.0
	26	- 18 925.4	- 18 925.4	0.0
	30	- 24 778.8	- 24 778.8	0.0
	34	- 35 849.6	- 35 849.6	0.0
	37	- 52 137.8	- 52 137.8	0.0
Upper	4	- 21 277.0	- 21 277.0	0.0
	8	- 37 336.7	- 37 336.6	0.0
	12	- 48 178.9	- 48 178.9	0.0
	16	- 53 803.7	- 53 803.7	0.0
	20	- 53 718.0	- 53 718.0	0.0
	24	- 48 864.6	- 48 864.6	0.0
	28	- 37 793.8	- 37 793.8	0.0
	32	- 21 505.6	- 21 505.6	0.0
Diagonal	1	- 23 593.5	- 23 593.5	0.0
	5	- 17 808.1	- 17 808.1	0.0
	9	- 12 022.7	- 12 022.7	0.0
	13	- 6 237.2	- 6 237.2	0.0
	17	- 247.9	- 247.9	0.0
	21	- 386.9	- 386.9	0.0
	25	- 6 490.7	- 6 490.7	0.0
	29	- 12 276.1	- 12 276.1	0.0
	33	- 18 061.6	- 18 061.6	0.0

TABLE 19 (Cont'd)

Truss-Member		SAP IV	EXACT	%-Error
Diagonal	36	- 23 846.9	- 23 846.9	0.0
Vertical	3	10 195.2	10 195.2	0.0
	7	7 695.2	7 695.2	0.0
	11	5 195.2	5 195.2	0.0
	15	2 695.2	2 695.2	0.0
	19	500.0	500.0	0.0
	23	2 804.8	2 804.8	0.0
	27	5 304.8	5 304.8	0.0
	31	7 804.8	7 804.8	0.0
	35	10 304.8	10 304.8	0.0

Displacement of Truss at Node 11.

SAP IV	EXACT	%-Error
.058"	.091"	56.9

B. Gravity Loads of Truss-Members Added by SAP IV and All Other Loads Concentrated at Nodes

a) Without Compressive Spring Force

TABLE 20

Truss Member		SAP IV	EXACT	%-Error
Lower	2,37	20 581.9	21 391.3	3.8
	6,34	36 451.6	37 565.2	3.0
	10,30	47 222.2	48 521.7	2.7
	14,26	52 872.0	54 260.9	2.6
	18,22	53 336.8	54 456.5	2.1
Upper	4,32	- 20 581.9	- 21 391.3	3.8
	8,28	- 36 451.6	- 37 565.2	3.0
	12,24	- 47 222.2	- 48 521.7	2.7
	16,20	- 52 872.0	- 54 260.9	2.6
Diag.	1,36	- 22 822.7	- 23 720.3	3.8
	5,33	- 17 597.4	- 17 934.8	1.9
	9,29	- 11 943.2	- 12 149.4	1.7
	13,25	- 6 264.9	- 6 363.9	1.6
	17,21	- 754.1	- 317.5	137.5
Vert.	3,35	9 464.1	10 250.0	7.7
	7,31	6 959.0	7 750.0	10.2
	11,27	4 453.9	5 250.0	15.2
	15,23	2 237.6	2 750.0	18.6
	19	719.9	500.0	44.0

b) With Compressive Spring-Force of 72800 lbs.

TABLE 21

Truss-Member		SAP IV	EXACT	%-Error
Lower	2,37	- 52 218.1	- 51 408.7	1.6
	6,34	- 36 348.4	- 35 234.8	3.2
	10,30	- 25 577.8	- 24 278.3	5.4
	14,26	- 19 927.9	- 18 539.3	7.5
	18,22	- 19 463.2	- 18 343.6	6.1
Upper	4,32	- 20 581.9	- 21 391.3	3.8
	8,28	- 36 451.6	- 37 565.2	3.0
	12,24	- 47 222.2	- 48 521.7	2.7
	16,20	- 52 872.0	- 54 260.9	2.7
Diag.	1,36	- 22 822.7	- 23 720.3	3.8
	5,33	- 17 597.4	- 17 934.8	1.9
	9,29	- 11 943.2	- 12 149.4	1.7
	13,25	- 6 264.9	- 6 363,9	1.6
	17,21	- 754.1	- 317.5	137.5
Vert.	3,35	9 464.1	10 250.0	7.7
	7,31	6 959.0	7 750.0	10.2
	11,27	4 453.9	5 250.0	15.2
	15,23	2 237.6	2 750.0	18.6
	19	719.9	500.0	44.0

c) With Compressive Spring Force of 72800 lbs.
and Internal Force of 1000 lbs.

TABLE 22

Truss-Member		SAP IV	EXACT	%-Error
Lower	2	- 52 332.4	- 51 522.9	1.6
	6	- 36 577.0	- 35 463.4	3.1
	10	- 25 920.7	- 24 621.1	5.3
	14	- 20 385.1	- 18 996.3	7.3
	18	- 19 963.2	- 18 772.7	6.3
	22	- 19 963.2	- 18 732.9	6.6
	26	- 20 470.9	- 19 081.9	7.3
	30	- 26 234.9	- 24 935.4]	5.2
	34	- 37 119.9	- 36 006.2	3.1
	37	- 53 103.8	- 52 294.4]	1.5
Upper	4	- 20 467.6	- 21 277.0	3.8
	8	- 36 222.9	- 37 336.6	3.0
	12	- 46 879.3	- 48 178.9	2.7
	16	- 52 414.9	- 53 803.7	2.6
	20	- 52 329.2	- 53 718.0	2.6
	24	- 47 565.0	- 48 864.6	2.6
	28	- 36 680.1	- 37 793.8	2.9
	32	- 20 696.2	- 21 505.6	3.8
Diag.	1	- 22 696.0	- 23 593.5	3.8
	5	- 17 470.7	- 17 808.1	1.9
	9	- 11 816.5	- 12 022.7	1.7
	13	- 6 138.2	- 6 327.2	1.6
	17	- 684.6	- 247.9	176.2
	21	- 823.7	- 386.9	112.9
	25	- 6 391.7	- 6 490.7	1.5

TABLE 22 (Cont'd)

Truss-Member		SAP IV	EXACT	%-Error
Diagonal	29	- 12 069.9	- 12 276.1	1.7
	33	- 17 724.2	- 18 061.6	1.9
	36	- 22 949.5	- 23 846.9	3.8
Vertical	3	9 409.4	10 195.2	7.7
	7	6 904.3	7 695.2	10.3
	11	4 399.2	5 195.2	15.3
	15	2 182.9	2 695.2	19.0
	19	719.9	500.0	44.0
	23	2 292.4	2 804.8	18.3
	27	4 508.7	5 304.8	15.0
	31	7 013.8	7 804.8	10.1

Displacement of Truss at Node 11:

SAP IV	EXACT	%-Error
.038"	.0899"	57.7

d) With Compressive Spring Force of 72800 lbs, Internal Force of 1000 lbs. and Internal Pressure of 150 psig.

TABLE 23

Truss-Member		SAP IV	EXACT	%-Error
Lower	2	- 52 175.8	- 51 366.4	1.6
	6	- 36 420.4	- 35 306.7	3.2
	10	- 25 764.1	- 24 464.5	5.3
	14	- 20 228.5	- 18 839.7	7.4
	18	- 19 806.6	- 18 686.9	6.0
	22	- 19 806.6	- 18 686.9	6.0
	26	- 20 314.3	- 18 925.4	7.3
	30	- 26 078.4	- 24 778.8	5.2
	34	- 36 963.3	- 35 849.6	3.1
	37	- 52 947.2	- 52 137.8	1.6
Upper	4	- 20 467.6	- 21 277.0	3.8
	8	- 36 222.9	- 37 336.6	3.0
	12	- 46 879.3	- 48 178.9	2.7
	16	- 52 414.9	- 53 803.7	2.6
	20	- 52 329.2	- 53 718.0	2.6
	24	- 47 565.0	- 48 864.6	2.7
	28	- 36 680.1	- 37 793.8	2.9
	32	- 20 696.2	- 21 505.6	3.8
Diag.	1	- 22 696.0	- 23 593.5	3.8
	5	- 17 470.7	- 17 808.1	1.9
	9	- 11 816.5	- 12 022.7	1.7
	13	- 6 138.2	- 6 237.2	1.6
	17	- 684.6	- 247.9	176.2

TABLE 23 (Cont'd)

Truss-Member	SAP IV	Exact	%-Error
Diagonal 21	-823.6	-386.9	112.9
25	-6391.7	-6490.7	1.5
29	-12096.9	-12276.1	1.5
33	-17724.2	-18061.6	1.9
36	-22949.5	-23846.9	3.8
Vertical 3	9409.4	10195.2	7.7
7	6904.3	95.2	10.3
11	4399.2	5195.2	15.3
15	2182.9	2695.2	19.0
19	719.9	500.0	44.0
23	2292.4	2804.8	18.3
27	4508.7	5304.8	15.0
31	7013.8	7804.8	10.1
35	9518.7	10304.8	7.6

Displacement of Truss at Node 11

SAP IV	Exact	% Error
.039"	.091"	57.1

X. CONCLUSIONS AND SUMMARY

A. All Loads Concentrated at Nodes.

The comparison of exact solutions versus SAP IV solutions shows complete agreement for all forces.

The displacement comparison shows an error of up to 57% as far as the SAP IV solution is concerned. The actual displacement, as measured with a telescope, is .090" when pressure vessel is pressurized.

This clearly shows that SAP IV gives perfect results as far as forces are concerned, but is in serious error for displacement solutions.

B. Gravity loads Distributed by SAP IV Program and All Other Loads Concentrated at Nodes

The comparison of exact versus SAP IV solutions shows an average error of between 2% to 4% for upper, lower and most diagonal members of the truss.

For the vertical members the error is between 8% to 44%. For two of the diagonal members, the error is 137.5%. The error of 2% to 4% for upper, lower and most diagonal members is within reason and acceptable as a viable solution. The error of between 8% to 44% for the vertical members is serious and not really acceptable. It is not clear which solutions are in error since strain-gage measurements are not available. However, since all members involved are manufactured from steel and the force differences are small, up to 800 lbs., the errors can be ignored.

Two diagonal members show a difference of 137.5%. Again, the members are made from steel and the force difference is of the order of 500 lbs., which leads to the conclusion that it can also be ignored for the purposes of this study.

The displacement comparison shows a %-error of 58% for the SAP IV solution. This is in serious error and not acceptable.

C. Summary

The overall conclusion that can be drawn from these calculations is that the glass-metal truss type bridge is structurally sound and that the precompression of 72800 lbs. is sufficient to insure stability of the structure.

The second conclusion is that both solution procedures, method of joints and finite element method, give acceptable solutions for this structure as far as forces are concerned. To show this more clearly, the force solution for upper, lower and diagonal members are listed once more beside the forces measured with strain-gages when the structure was under a precompression of 68000 lbs.

TABLE 24

Truss-Member		SAP IV	EXACT	Strain-Gage
Lower	2	- 52 218	- 51 408	- 46 600
	6	- 36 348	- 35 235	- 30 300
	10	- 25 578	- 24 278	- 27 500
	14	- 19 928	- 18 539	- 23 000
Upper	4	20 582	21 391	12 600
	8	36 452	37 565	24 700
	12	47 222	48 522	31 600
Diagonal	1	- 22 823	- 23 720	- 17 200
	5	- 17 597	- 17 935	- 16 800
	9	- 11 943	- 12 149	- 10 300
	13	- 6 265	- 6 364	- 8 000

It is seen that the force pattern is almost linear for all solutions. The strain-gage measurements are somewhat erratic but this is clearly due to gage inaccuracy and to variations in construction of members.

Overall, however, the distribution of forces is close to that for an ideal truss.

The third conclusion is that displacement solutions from SAP IV are not acceptable, while the solutions from the method of joints are basically 100% correct. Any further calculations for displacement should therefore be based on the method of joints.

For a possible explanation of the SAP lV displacement solutions see appendix Xll D.

TABLE 25

Physical Data of Truss-Members

Member Number	Cross-Sectional Area (in.^{2})	Length (in.)	Density
1	36	106.452	.08044
2	64	96.0	.08044
3	9	46.0	.283
4	48	96.0	.08044
5	36	106.452	.08044
6	48	96.0	.08044
7	9	46.0	.283
8	64	96.0	.08044
9	36	106.452	.08044
10	48	96.0	.08044
11	9	46.0	.283
12	64	96.0	.08044
13	24	106.452	.08044
14	48	96.0	.08044
15	9	46.0	.283
16	12	36.0	.283
17	15	58.412	.283
18	12	36.0	.283
19	15	46.0	.283
20	12	36.0	.283
21	15	58.412	.283
22	12	36.0	.283
23	9	46.0	.283
24	64	96.0	.08044
25	24	106.452	.08044
26	48	96.0	.08044
27	9	46.0	.283
28	64	96.0	.08044
29	36	106.452	.08044

TABLE 25 (Cont'd)

Member Number	Cross-Sectional Area (in.^{2})	Length (in.)	Density
30	48	96.0	.08044
31	9	46.0	.283
32	48	96.0	.08044
33	36	106.452	.08044
34	48	96.0	.08044
35	9	46.0	.283
36	36	106.452	.08044
37	64	96.0	.08044

TABLE 26

Physical Constants

Spring Constant: k_s = 1800 lb./in.
Tank Expansion Coefficient: .0058 in./10 psi
Modulus of Elasticity of Glass: 5×10^6 lb./in.^{2}
Modulus of Elasticity of Steel: 30×10^6 lb./in.^{2}
Spring Force on Truss Structure: 72800 lbs.

XI. References

1. Energy Principles in Structural Mechanics, Theodore R. Tauchert, McGaw-Hill Book Company, N.Y. 1974.
2. Elementary Theory of Structures, Yuan-Yu Hsien, Prentice-Hall, Inc., Englewood Cliffs, N.J. 1970.
3. Introduction to Finite Element Analysis, Harold C. Martin, Graham F. Carey, McGraw-Hill Book Company, N.Y. 1973.
4. SAP IV

   A structural analysis program for static and dynamic response of linear systems, by Klaus-Jürgen Bathe, Edward L. Wilson, Fred E. Peterson, College of Engineering, University of California, Berkeley, California.

A. Outline of SAP IV program

1. INTRODUCTION

The development of an effective computer program for structural analysis requires a knowledge of three scientific disciplines -- structural mechanics, numerical analysis and computer application. The development of accurate and efficient structural elements requires a modern background in structural mechanics. The efficiency of a program depends largely on the numerical techniques employed and on their effective computer implementation. With regard to programming techniques, an optimum allocation of high and low speed storage is necessary.

A most important aspect of a general purpose computer program is, however, the ease with which it can be modified, extended and updated; otherwise, it may very well be that the program is obsolete within a few years after completion. This is because new structural elements are developed, better numerical procedures are available, or new computer equipment which requires new coding techniques is produced.

The structural analysis program SAP was designed to be modified and extended by the user. Additional options and new elements may easily be added. The program has the capacity to analyze very large three-dimensional systems; however, there is no loss in efficiency in the solution of smaller problems. Also, from the complete program, smaller special purpose programs can easily be assembled by simply using only those subroutines which are actually needed in the execution. This makes the program particularly usable on small size computers.

The current program version SAP IV for the static and dynamic analysis of linear structural systems is the result of several years' research and development experience. The program has proven to be a very flexible and efficient analysis tool. The program is coded in FORTRAN IV and operates without modifications on the CDC 6400, 6600 and 7600 computers. The first version of program SAP was published in September 1970 [28]. An improved static analysis program, namely SOLID SAP, or SAP II, was presented in 1971 [29]. Work was then started on a new static and dynamic analysis program. The program SAP III for static and dynamic analysis was released towards the end of 1972, but only to those agencies which supported our research. In relation to SAP III, the current version SAP IV has improvements throughout, and in particular has available a new variable-number-nodes thick shell and three-dimensional element, and out-of-core direct integration for time history analysis.

The structural systems to be analyzed may be composed of combinations of a number of different structural elements. The program presently contains the following element types:

(a) three-dimensional truss element,

(b) three-dimensional beam element,

(c) plane stress and plane strain element,

(d) two-dimensional axisymmetric solid,

(e) three-dimensional solid,

(f) variable-number-nodes thick shell and three-dimensional element,

(g) thin plate or thin shell element,

(h) boundary element,

(i) pipe element (tangent and bend).

These structural elements can be used in a static or dynamic analysis. The capacity of the program depends mainly on the total number of nodal points in the system, the number of eigenvalues needed in the dynamic analysis and the computer used. There is practically no restriction on the number of elements used, the number of load cases or the order and bandwidth of the stiffness matrix. Each nodal point in the system can have from zero to six displacement degrees of freedom. The element stiffness and mass matrices are assembled in condensed form; therefore, the program is equally efficient in the analysis of one-, two- or three-dimensional systems.

The formation of the structure matrices is carried out in the same way in a static or dynamic analysis. The static analysis is continued by solving the equations of equilibrium followed by the computation of element stresses. In a dynamic analysis the choice is between

1. frequency calculations only,
2. frequency calculations followed by response history analysis,
3. frequency calculations followed by response spectrum analysis,
4. response history analysis by direct integration.

To obtain the frequencies and vibration mode shapes solution routines are used which calculate the required eigenvalues and eigenvectors directly without a transformation of the structure stiffness matrix and mass matrix to a reduced form. In the direct integration an unconditionally stable integration scheme is used, which also operates on the original structure stiffness matrix and mass matrix. This way the program operation and necessary input data for a dynamic analysis is a simple addition to what is needed for a static analysis.

2. THE EQUILIBRIUM EQUATIONS FOR COMPLEX STRUCTURAL SYSTEMS

2.1 Element to Structure Matrices

The nodal point equilibrium equations for a linear system of structural elements can be derived by several different approaches [1] [2] [9] [15] [23] [34]. All methods yield a set of linear equations of the following form

$$M\ddot{u} + C\dot{u} + Ku = R \tag{1}$$

where M is the mass matrix, C is the damping matrix and K is the stiffness matrix of the element assemblage; the vectors u, $\dot{u}$, $\ddot{u}$ and R are the nodal displacements, velocities, accelerations and generalized loads, respectively. The structure matrices are formed by direct addition of the element matrices; for example

$$K = \sum K_m \tag{2}$$

where K_m is the stiffness matrix of the m'th element. Although K_m is formally of the same order as K, only those terms in K_m which pertain to the element degrees of freedom are nonzero. The addition of the element matrices can therefore be performed by using the element matrices in compact form together with identification arrays which relate element to structure degrees of freedom. The algorithm used in the program is described in Section 3.3.

In the program the structure stiffness matrix and a diagonal mass matrix are assembled. Therefore, a lumped mass analysis is assumed, where the structure mass is the sum of the individual element mass matrices plus additional concentrated masses which are specified at selected degrees of freedom. The damping is assumed to be proportional and is specified in form of a modal damping factor. The assumptions used in lumped mass analyses and in the use of proportional damping have been discussed at various occasions [9] [11] [17] [33].

2.2 Boundary Conditions

If a displacement component is zero, the corresponding equation is not retained in the structure equilibrium equations, Eq. (1), and the corresponding element stiffness and mass terms are disregarded. If a non-zero displacement is to be specified at a degree of freedom i, say $u_i = x$, the equation

$$k\, u_i = k\, x \tag{3}$$

is added into Eq. (1), where $k \gg k_{ii}$. Therefore, the solution of Eq. (1) must give $u_i = x$. Physically, this can be interpreted as adding at the degree of freedom "i" a spring of large stiffness k and specifying a load which, because of the relatively flexible structure at this degree of freedom, produces the required displacement x.

3. PROGRAM ORGANIZATION FOR CALCULATION OF THE STRUCTURE STIFFNESS MATRIX AND MASS MATRIX

The calculation of the structure stiffness matrix and mass matrix is accomplished in three distinct phases:

1. The nodal point input data is read and generated by the program. In this phase the equation numbers for the active degrees of freedom at each nodal point are established.
2. The element stiffness and mass matrices are calculated together with their connection arrays; the arrays are stored in sequence on tape (or other low-speed storage).
3. The structure stiffness matrix and mass matrix are formed by addition of the element matrices and stored in block form on tape.

It need be noted that these basic steps are indenendent of the element type used and are the same for either a static or dynamic analysis.

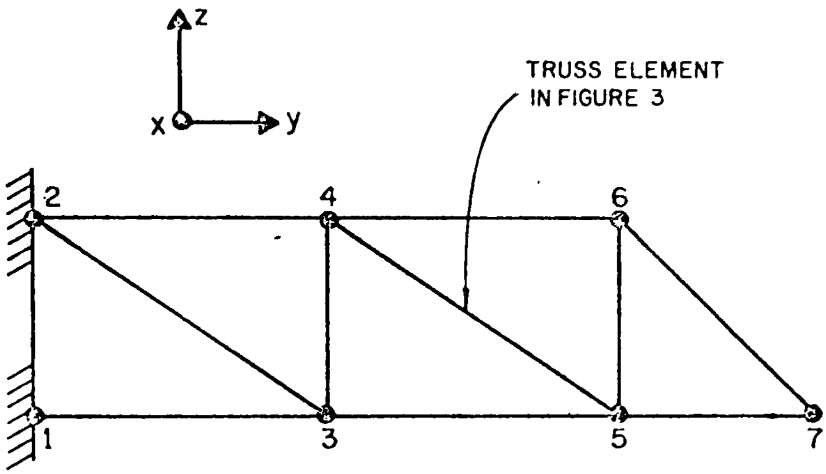
3.1 Nodal Point Input Data and Degrees of Freedom

The capacity of the program is controlled by the number of nodal points of the structural system. For each nodal point six boundary condition codes (stored in the array ID), three coordinates (stored in the arrays X,Y,Z) and the nodal point temperatures (stored in the array T) are required (generation capability is provided). All nodal point data is retained in high speed storage during the formation of the element stiffness and mass matrices. Since the required high speed storage for the element subroutines is relatively small, the minimum required storage for a given problem is a little larger than ten times the number of nodal points in the system.

It need be noted that the user should allow only those degrees of freedom which are compatible with the elements connected to a nodal point. The program always deals with six possible degrees of freedom at each nodal point, and all non-active degrees of freedom should be deleted, so as to decrease the order of the structure matrices. Specifically, a "1" in the ID array denotes that no equation shall be associated with the degree of freedom, whereas a "0" indicates that this is an active degree of freedom. Figure 1 shows for the simple truss structure the ID array as it was read and/or generated by the program. Once the complete ID and X,Y,Z arrays have been obtained, equation numbers are associated with all active degrees of freedom, i.e., the zeroes in the ID array are replaced by corresponding equation numbers, and each one is replaced by a zero, as shown in Fig. 2 for the simple truss example.

3.2 Element Mass and Stiffness Calculations

With the coordinates of all nodal points known and the equation numbers of the degrees of freedom having been established, the stiffness, mass and stress-displacement transformation matrices for each structural element in the system are calculated. As pointed out earlier, little additional high-speed storage is required for this phase since these matrices are formed and placed on tape storage at the same time as the element properties are read. Together with the matrices pertaining to the element, the corresponding element connection array, vector LM, is written on tape. The vector LM is established



NODAL POINT LAYOUT OF TRUSS

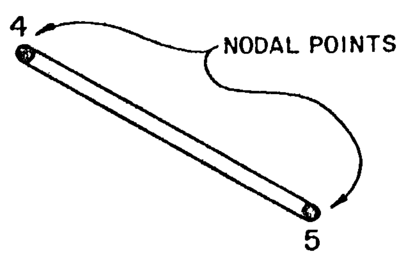
$$
\begin{array}{cc}
 & \begin{array}{cccccc} 1 & 2 & 3 & 4 & 5 & 6 \end{array} \leftarrow \text{DEGREES OF FREEDOM} \\
ID = \begin{array}{c} 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \\ 7 \end{array} &
\begin{bmatrix}
1 & 1 & 1 & 1 & 1 & 1 \\
1 & 1 & 1 & 1 & 1 & 1 \\
1 & 0 & 0 & 1 & 1 & 1 \\
1 & 0 & 0 & 1 & 1 & 1 \\
1 & 0 & 0 & 1 & 1 & 1 \\
1 & 0 & 0 & 1 & 1 & 1 \\
1 & 0 & 0 & 1 & 1 & 1
\end{bmatrix}
\end{array}
$$

NODAL POINT
NUMBERS

NODAL POINT LAYOUT OF TRUSS-EXAMPLE AND ID-ARRAY AS READ AND/OR GENERATED

$$ID = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 2 & 0 & 0 & 0 \\ 0 & 3 & 4 & 0 & 0 & 0 \\ 0 & 5 & 6 & 0 & 0 & 0 \\ 0 & 7 & 8 & 0 & 0 & 0 \\ 0 & 9 & 10 & 0 & 0 & 0 \end{bmatrix}$$

ID ARRAY OF TRUSS-EXAMPLE AFTER ALLOCATION OF EQUATION NUMBERS TO ACTIVE DEGREES OF FREEDOM



$$LM = \begin{bmatrix} 0 \\ 3 \\ 4 \\ 0 \\ 5 \\ 6 \end{bmatrix}$$

CONNECTION ARRAY (VECTOR LM) FOR A TYPICAL ELEMENT OF THE TRUSS-EXAMPLE from the ID matrix and the specified structure nodal points pertaining to the element. The connection array for a typical element of the truss element is shown in Fig. 3.

The element matrices are calculated in groups, i.e., always all elements in one group together, thus calling the corresponding element subroutine only once for each element group. After all element matrices have been established, the ID and X,Y,Z arrays are not needed any more, and the corresponding storage area is used for the formation of the structure matrices and later for the solution of the equations of equilibrium.

3.3 Formation of Structure Stiffness and Mass

The stiffness matrix and mass matrix of the structure are formed in blocks, as shown in Fig. 4 for the truss-example. The number of equations per block depends on the available high speed storage and is calculated in the program as indicated in Fig. 5. It is noted that on reasonable size computers very large systems can be analyzed for static and dynamic response. With the number of equations per block known, the stiffness and mass matrix are assembled two blocks at a time by direct addition of the element matrices. In this process it is necessary to pass through the element matrices which are stored on tape. In order to minimize tape reading, in each pass element matrices which pertain to the next several blocks are written on another tape. This way the tape reading necessary for the formation of these blocks is reduced significantly.

A flow diagram of the program organization for the calculation of the structure stiffness matrix and mass matrix is shown in Fig. 6.

0 = ZERO ELEMENT

X = NONZERO ELEMENT

$$\begin{bmatrix} X & X & 0 & 0 & X & & & & & \\ & X & 0 & X & 0 & 0 & & \text{ZEROS} & & \\ & & X & X & X & X & X & & & \\ & & & X & X & X & 0 & 0 & & \\ & & & & X & X & 0 & 0 & X & \\ & & & & & X & 0 & X & 0 & 0 \\ & \text{SYMMETRIC} & & & & & X & X & X & X \\ & & & & & & & X & X & X \\ & & & & & & & & X & X \\ & & & & & & & & & X \end{bmatrix} \qquad \begin{bmatrix} X \\ X \\ X \\ X \\ X \\ X \\ X \\ X \\ X \\ X \end{bmatrix}$$

STIFFNESS MATRIX

MASS MATRIX

ACTUAL STRUCTURE MATRICES

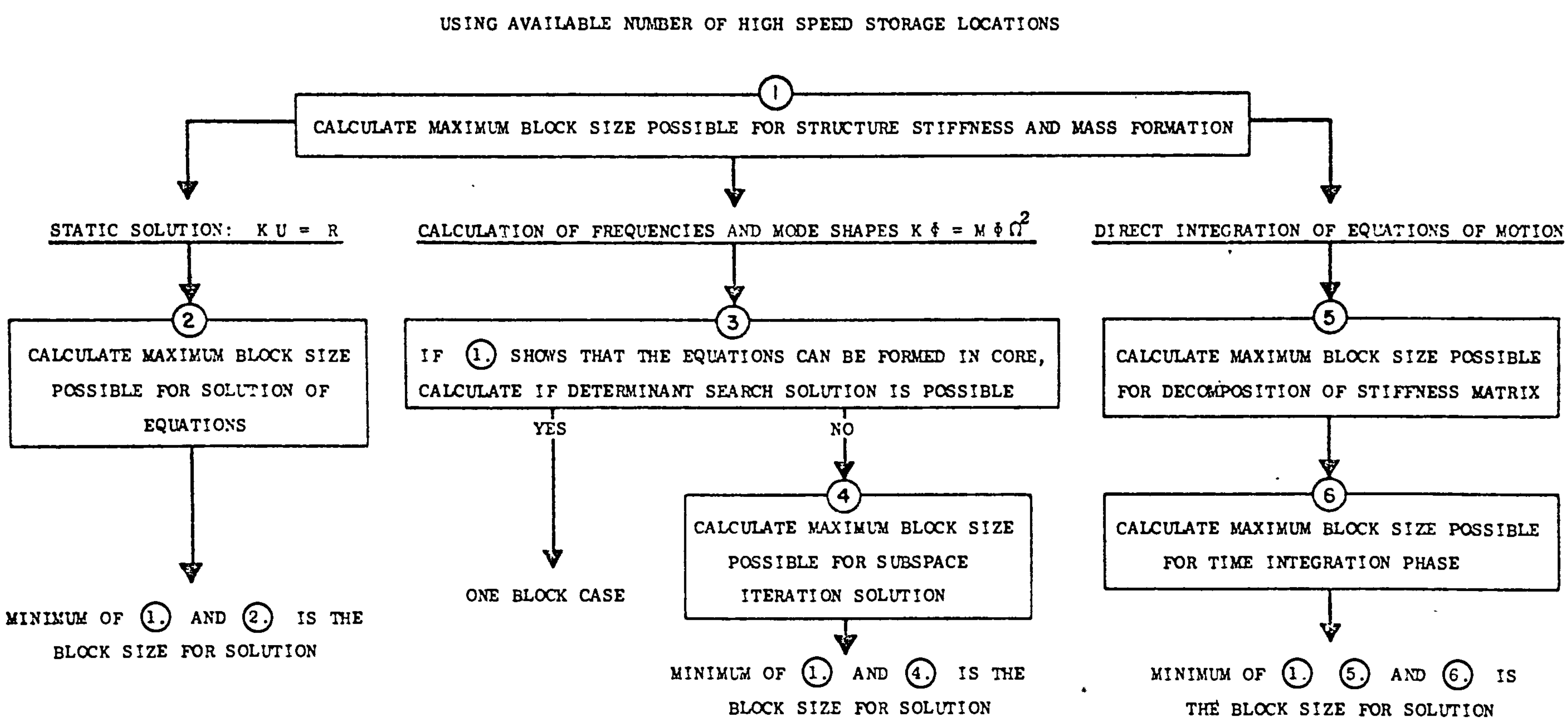
$$\begin{array}{c} \text{BLOCK 1} \\ \\ \text{BLOCK 2} \\ \\ \text{BLOCK 3} \\ \\ \text{BLOCK 4} \end{array} \left[\begin{array}{ccccc} X & X & 0 & 0 & X \\ X & 0 & X & 0 & 0 \\ X & X & X & X & X \\ \hline X & X & X & 0 & 0 \\ X & X & 0 & 0 & X \\ X & 0 & X & 0 & 0 \\ \hline X & X & X & X & \\ X & X & X & & \\ X & X & & \text{ZEROS} & \\ \hline X & & & & \end{array}\right] \qquad \left[\begin{array}{c} X \\ X \\ X \\ \hline X \\ X \\ X \\ \hline X \\ X \\ X \\ \hline X \\ 0 \\ 0 \end{array}\right]$$

STIFFNESS MATRIX

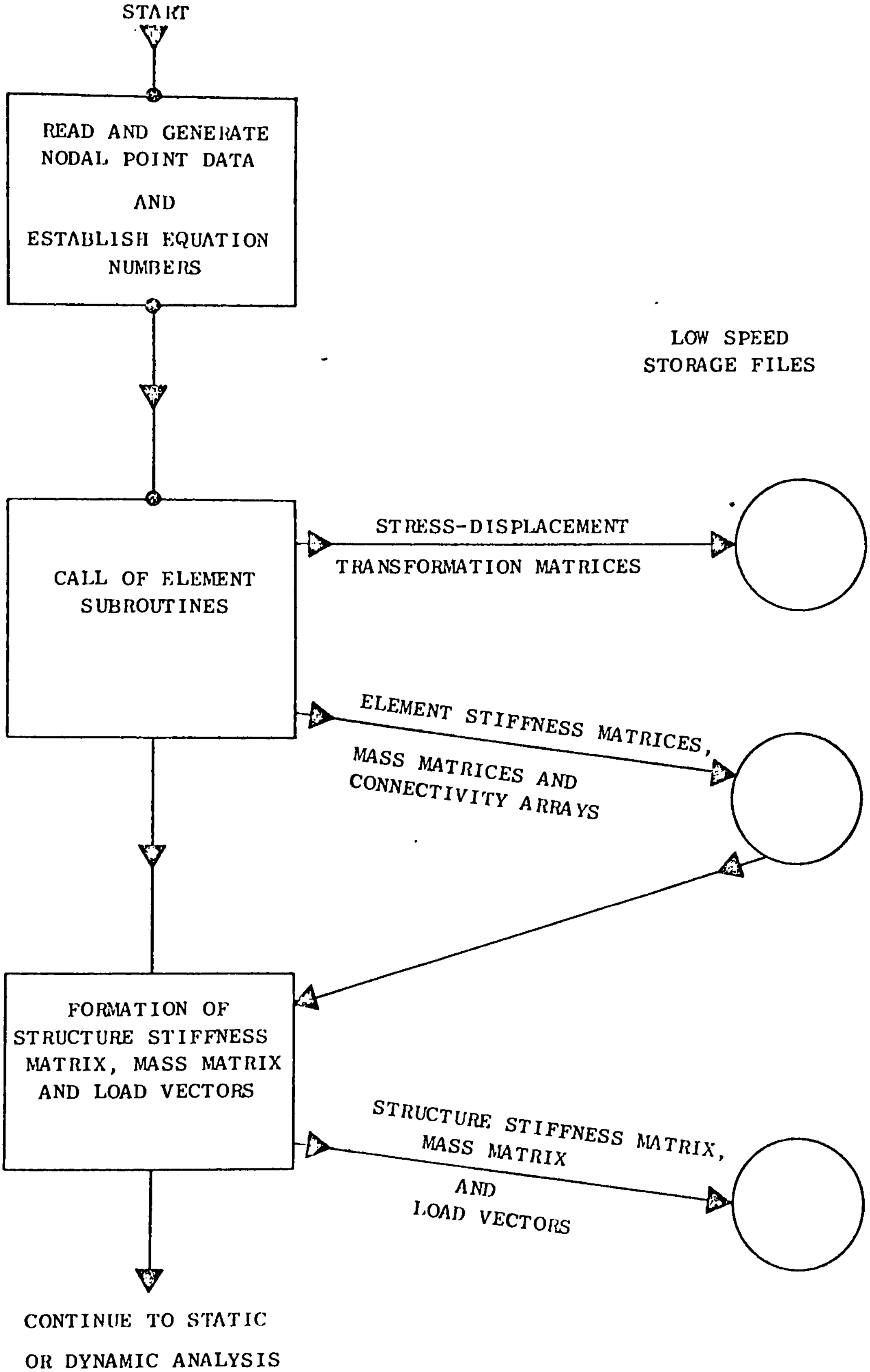
MASS MATRIX

BLOCK STORAGE OF STRUCTURE MATRICES

STORAGE OF STIFFNESS MATRIX AND MASS MATRIX ON TAPE



FLOWCHART SHOWING CALCULATION OF
NUMBER OF EQUATIONS IN A BLOCK



FLOWCHART FOR CALCULATION OF STRUCTURE STIFFNESS MATRIX AND MASS MATRIX

With the matrices stored in block form on tape either a static or a dynamic analysis can now be carried out.

4. THE ELEMENT LIBRARY

The element library of SAP IV consists of eight different element types. These elements can be used in either a static or dynamic analysis. They are shown in Fig. 7 and are briefly described below.

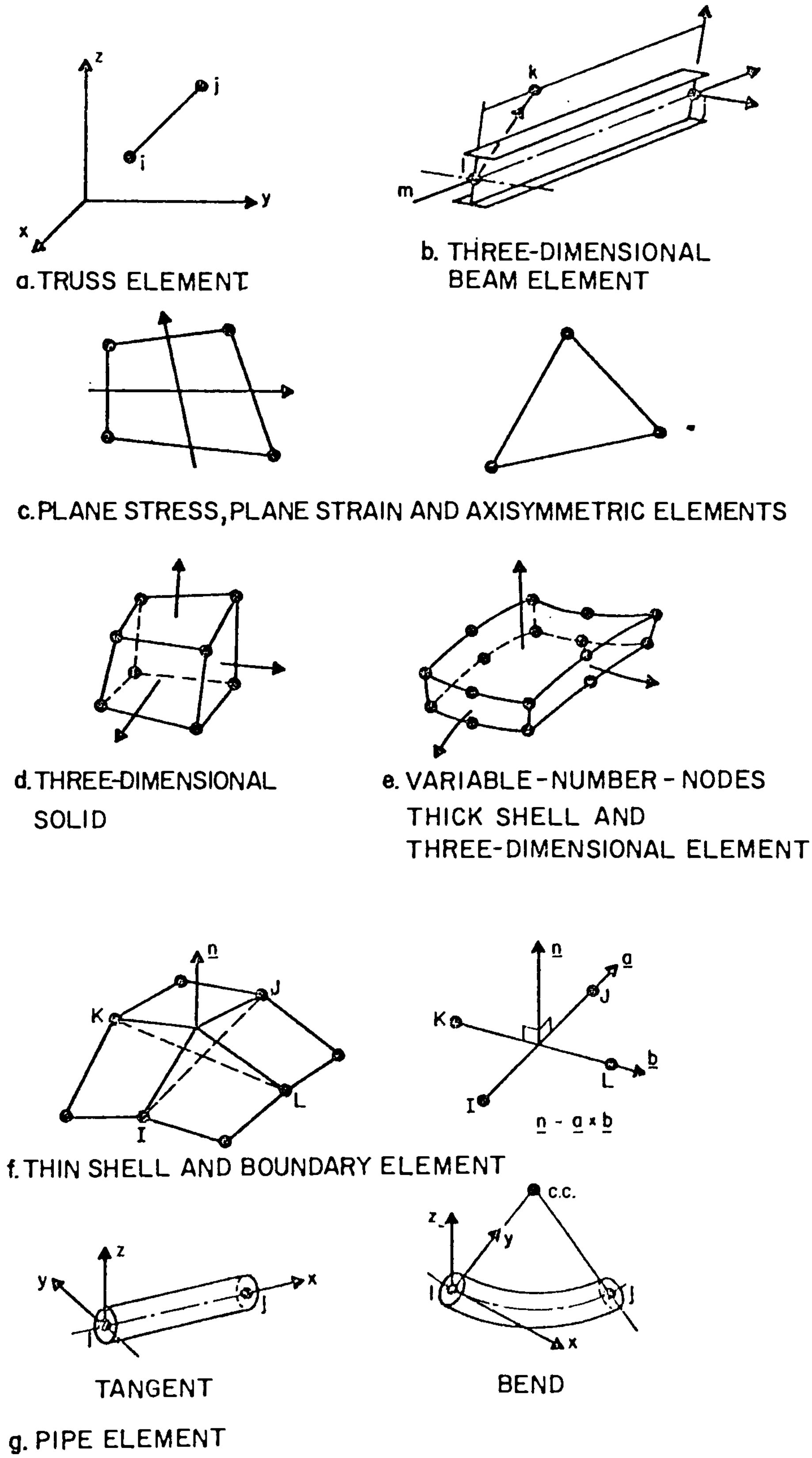
4.1 Three Dimensional Truss Element

The derivation of the truss element stiffness is given in Refs. [23] [29]. The element can be subjected to a uniform temperature change.

4.2 Three-Dimensional Beam Element

The beam element included in the program considers torsion, bending about two axes, axial and shearing deformations. The element is prismatic. The development of its stiffness properties is standard and is given in Ref. [23]. Inertia loading in three directions and specified fixed-end-forces form the element load cases. Forces (axial and shear) and moments (bending and torsion) are calculated in the beam local co-ordinate system.

A typical beam element is shown in Fig. 7b. A plane which defines the principal bending axis of the beam is specified by the plane i, j, k. Only the geometry of nodal point k is needed; therefore, no additional degrees of freedom for nodal point k are used in the computer program. A unique option of the beam member is that the ends of the beam can be geometrically constrained to a master node. Slave degrees of freedom at the end of the beam are eliminated from the formulation and replaced by the transformed degrees of freedom of the master node [18] [29]. This technique reduces the total number of joint equilibrium



ELEMENT LIBRARY OF SAP IV

equations in the system (while possibly increasing the bandwidth) and greatly reduces the possibility of numerical sensitivities in many types of structures. Also, the method can be used to specify rigid floor diaphragms in building analysis.

4.3 Plane Stress, Plane Strain and Axisymmetric Elements

A plane stress quadrilateral (or triangular) element with orthotropic material properties is available. Each plane stress element may be of different thickness and may be located in an arbitrary plane with respect to the three-dimensional coordinate system. The plane strain and axisymmetric elements are restricted to the y-z plane. Gravity, inertia and temperature loadings may be considered. Stresses may be computed at the center of the element and at the center of each side. The element is based on an isoparametric formulation [19] [34]. Incompatible displacement modes can be included in order to improve the bending properties of the element [26] [29] [32].

4.4 Three-Dimensional Solid Element

A general eight nodal point "brick" element, with three translational degrees of freedom per nodal point can be used, Fig. 7d. Isotropic material properties are assumed and element loading consists of temperature, surface pressure and inertia loads in three directions. Stresses (six components) may be computed at the center of the element and at the center of each face. The element employs incompatible modes, which can be very effective if rectangular elements are used [26].

4.5 Variable-Number-Nodes Thick Shell and Three-Dimensional Element

A general three-dimensional isoparametric or subparametric element which may have from 8 to 21 nodes can be used for three-dimensional or thick shell analysis, Fig. 7e [7] [8]. General orthotropic material properties can be assigned to the element. The loading may consist of applied surface pressure, hydrostatic loads, inertia loads in three directions, and thermal loads. Six global stresses are output at up to seven locations within an element.

4.6 Thin Plate and Shell Element

The thin shell element available in the program is a quadrilateral of arbitrary geometry formed from four compatible triangles. The bending and plane stress properties of the element are described in references [12] [14]. The shell element uses the constant strain triangle and the LCCT9 element to represent the membrane and bending behavior, respectively. The central node is located at the average of the coordinates of the four corner nodes. The element has six interior degrees of freedom which are eliminated at the element level prior to assembly; therefore, the resulting quadrilateral element has twenty-four degrees of freedom, i.e., six degrees of freedom per node in the global coordinate system.

In the analysis of flat plates the stiffness associated with the rotation normal to the shell surface is not defined; therefore, the rotation normal degree of freedom must not be included in the analysis. For curved shells, the normal rotation need be included as an extra degree of freedom. In case the curvature is very small, the degree of freedom should be restrained by the addition of a "Boundary Element" with a small normal rotational stiffness, say of less or about 10% of the element bending stiffness [13] [34].

4.7 Boundary Element

The boundary element, shown in Fig. 7f, can be used for the following:

1. in the idealization of an external elastic support at a node;
2. in the idealization of an inclined roller support;
3. to specify a displacement, or
4. to eliminate the numerical difficulty associated with the 'sixth' degree of freedom in the analysis of nearly flat shells.

The element is one-dimensional with an axial or torsional stiffness. The element stiffness coefficients are added directly to the total stiffness matrix (see Section 2.2).

4.8 Pipe Element

The pipe element (Fig. 7g) can represent a straight segment (tangent) or a circularly curved segment (bend); both elements require a uniform section and uniform material properties. Elements can be directed arbitrarily in space. The member stiffness matrices account for bending, torsional, axial and shearing deformations. In addition, the effect of internal pressure on the stiffness of curved pipe elements is considered.

The types of structure loads contributed by the pipe elements include gravity loading in the global directions, and loads due to thermal distortions and deformations induced by internal pressure. Forces and moments acting at the member ends (i,j) and at the center of each bend are calculated in coordinate systems aligned with the member's cross section.

The pipe element stiffness matrix is formed by first evaluating the flexibility matrix corresponding to the six degrees of freedom at end j as given by Poley [22]. With the corresponding stiffness matrix, the equilibrium transformations outlined by Hall et al [16] are used to form the complete element stiffness matrix. Distortions due to element loads are premultiplied by the stiffness matrix to compute restrained nodal forces due to thermal, pressure or gravity loads.

5. STATIC ANALYSIS

A static analysis involves the solution of the equilibrium equations

$$K u = R \tag{4}$$

followed by the calculation of element stresses.

5.1 Solution of Equilibrium Equations

The load vectors R have been assembled at the same time as the structure stiffness matrix and mass matrix were formed. The solution of the equations is obtained using the large capacity linear equation solver SESOL [31]. This subroutine uses Gauss elimination on the positive-definite symmetrical system of equations. The algorithm performs a minimum number of operations; i.e. there are no operations with zero elements. In the program, the $L^T DL$ decomposition of K is used, hence Eq. (4) can be written as

$$L^T v = R \tag{5}$$

and

$$v = DLu \tag{6}$$

where the solution for v in Eq. (5) is obtained by a reduction of the load vectors; the displacement vectors u are then calculated by a back-substitution.

In the solution, the load vectors are reduced at the same time as K is decomposed. In all operations it is necessary to have at any one time the required matrix elements in high-speed storage. In the reduction, two blocks are in high speed storage (as was also the case in the formation of the stiffness matrix and mass matrix), i.e., the "leading" block, which finally stores the elements of L and D, and in succession those blocks which are affected by the decomposition of the "leading" block. Table 1 gives some typical solution times.

5.2 Evaluation of Element Stresses

After the nodal point displacements have been evaluated, sequentually the element stress-displacement matrices are read from low speed storage and the element stresses are calculated.

B. DATA INPUT TO SAP IV

I. HEADING CARD (12A6)

notes	columns	variable	entry
(1)	1 - 72	HED(12)	Enter the heading information to be printed with the output

NOTES:

(1) Begin each new data case with a new heading card.

II. MASTER CONTROL CARD (8I5)

notes	columns	variable	entry
(1)	1 - 5	NUMNP	Total number of nodal points (joints) in the model
(2)	6 - 10	NELTYP	Number of element groups
(3)	11 - 15	LL	Number of structure load cases; GE.1; static analysis EQ.0; dynamic analysis
(4)	16 - 20	NF	Number of frequencies to be found in the eigenvalue solution; EQ.0; static analysis GE.1; dynamic analysis
(5)	21 - 25	NDYN	Analysis type code: EQ.0; static analysis EQ.1; eigenvalue/vector solution EQ.2; forced dynamic response by mode superposition EQ.3; response spectrum analysis EQ.4; direct step-by-step integration
(6)	26 - 30	MODEX	Program execution mode: EQ.0; problem solution EQ.1; data check only
(7)	31 - 35	NAD	Total number of vectors to be used in a SUBSPACE INTERATION solution for eigenvalues/vectors: EQ.0; default set to: MIN{2*NF,NF+8}
(8)	36 - 40	KEQB	Number of degrees of freedom (equations) per block of storage: EQ.0; calculated automatically by the program

NOTES.

(1) Nodes are labeled with integers ranging from "1" to the total number of nodes in the system, "NUMNP" The program exits with no diagnostic message if NUMNP is zero (0). Thus, two blank cards are used to end the last data case in a run; i.e.. one blank heading card (Section I) and one blank card for this section.

(2) For each different element type (TRUSS, BEAM, etc.) a new element group need be defined. Elements within groups are assigned integer labels ranging from "1" to the total number of elements in the group. Element groups are input in Section IV, below.

II. MASTER CONTROL CARD (continued)

Element numbering must begin with one (1) in each different group. It is possible to use more than one group for an element type. For example, all columns (vertical beams) of a building may be considered one group and the girders (horizontal beams) may be considered another group.

(3) At least one (1) load condition must be specified for a static (NDYN.EQ.0) analysis. If the data case calls for one of the dynamic analysis options (NDYN.EQ.1, 2, 3, or 4), no load cases can be requested (i.e., LL is input as "0"). The program always processes Sections V (Concentrated Load/Mass Data) and VI (Element Load Multipliers) and expects to read some data. For the case of a dynamic analysis (NDYN.GE.1) only mass coefficients can be input in Section V, and one (1) blank element load multiplier card is expected in Section VI.

(4) For a static analysis, NF.EQ.0. If NDYN.EQ.1, 2 or 3, the lowest NF eigenvalues are determined by the program. Note that a dynamic solution may be re-started after eigenvalue extraction (providing a previous eigenvalue solution for the model was saved on tape as described in Appendix A). NF for the original and re-start runs must be the same.

(5) If NDYN.EQ.2 or NDYN.EQ.3 the program first solves for NF eigenvalues/vectors and then performs the forced response solution (or the response spectrum analysis). Thus, the program expects to read the control card governing the eigensolution (Section VII.A) before reading data in either Sections VII.B or VII.C. For the case NDYN.EQ.1, the program solves for NF eigenvalues/vectors, prints the results and proceeds to the next data case. The results for the eigenvalue solution phase (NDYN.EQ.1) may be saved for later use in automatic re-start (Appendix A lists the control cards that are required to affect this save operation), i.e. a dynamic solution may be restarted without repeating the solution for modes and frequencies. If this data case is a re-start job, set NDYN.EQ.-2 for a forced response solution, or set NDYN.EQ.-3 for a response spectrum analysis. Note that the solution may be re-started a multiple of times (to run different ground spectra or different time-dependent forcing functions) because the program does not destroy the contents of the re-start tape.

If NDYN.EQ.4 the program performs the response solution by direct step-by-step integration and no eigenvalue solution control card should be provided.

II. MASTER CONTROL CARD (continued)

(6) In the data-check-only mode (MODEX.EQ.1), the program writes only one file, "TAPE8", and this file may be saved for use as input to special purpose programs such as mesh plotters, etc. TAPE8 contains all data input in its completely generated form. If MODEX.EQ.1, most of the expensive calculations required during normal (MODEX.EQ.0) execution are passed. TAPE8, however, is not written during normal problem solution.

Note that a negative value for NDYN ("-2" or "-3"), when executing in the data-check-only mode, does not cause the program to read the re-start tape which contains the eigensolution information; instead, the program jumps directly from this card to Section VII.B (or Section VII.C) and continues reading and checking data cards without performing the solution.

(7) If the program is to solve for eigenvalues using the SUBSPACE ITERATION algorithm, the entry in cc 31-35 can be used to change the total number of iteration vectors to be used from the default minimum of 2*NF or NF+8 (whichever is smaller) to the value "NAD". The effect of increasing NAD over the default value is to accelerate convergence in the calculations for the lowest NF eigenvalues. NAD is principally a program testing parameter and should normally be left blank.

(8) KEQB is a program testing parameter which allows the user to test multiple equation block solutions using small data cases which would otherwise be one block problems. KEQB is normally left blank.

III. NODAL POINT DATA (A1,I4,6I5,3F10.0,I5,F10.0)

notes	columns	variable	entry
(1)	1	CT	Symbol describing coordinate system for this node; EQ. ; (blank) cartesian (X,Y,Z) EQ.C; cylindrical (R,Y,θ)
(2)	2 - 5	N	Node number
(3)	6 - 10	IX(N,1)	X-translation boundary condition code
	11 - 15	IX(N,2)	Y-translation boundary condition code
	16 - 20	IX(N,3)	Z-translation boundary condition code
	21 - 25	IX(N,4)	X-rotation boundary condition code
	26 - 30	IX(N,5)	Y-rotation boundary condition code
	31 - 35	IX(N,6)	Z-rotation boundary condition code EQ.0; free (loads allowed) EQ 1; fixed (no load allowed) GT.1; master node number (beam nodes only)
(4)	36 - 45	X(N)	X (or R) -ordinate
	46 - 55	Y(N)	Y -ordinate
	56 - 65	Z(N)	Z (or θ) -ordinate (degrees)
(5)	66 - 70	KN	Node number increment
(6)	71 - 80	T(N)	Nodal temperature

NOTES.

(1) A special cylindrical coordinate system is allowed for the global description of nodal point locations. If a "C" is entered in card column one (1), then the entries given in cc 36-65 are taken to be references to a global (R,Y,θ) system rather than to the standard (X,Y,Z) system. The program converts cylindrical coordinate references to cartesian coordinates using the formulae:

$$X = R \sin\theta$$
$$Y = Y,$$
$$Z = R \cos\theta$$

Cylindrical coordinate input is merely a user convenience for locating nodes in the standard (X,Y,Z) system, and no other references to the cylindrical system are implied; i.e., boundary condition specifications, output displacement components, etc. are referenced to the (X,Y,Z) system.

(2) Nodal point data must be defined for all (NUMNP) nodes. Node data may be input directly (i.e., each node on its own individual card) or the generation option may be used if applicable (see note 5, below).

III. NODAL POINT DATA (continued)

Admissible nodal point numbers range from "1" to the total number of nodes "NUMNP". Illegal references N.LE.0 or N.GT.NUMNP.

(3) Boundary condition codes can only be assigned the following values (M = 1,2,...,6):

IX(N,M) = 0;	unspecified (free) displacement (or rotation) component
IX(N,M) = 1;	deleted (fixed) displacement (or rotation) component
IX(N,M) = K;	node number "K" (1 < K .. NUMNP and K ≠ N) is the "master" node to which the Mth degree of freedom at node "N" is a "slave"

An unspecified (IX(N,M) = 0) degree of freedom is free to translate or rotate as the solution dictates. Concentrated forces (or moments) may be applied (Section V, below) in this degree of freedom. One (1) system equilibrium equation is required for each unspecified degree of freedom in the model. The maximum number of equilibrium equations is always less than six (6) times the total number of nodes in the model.

Deleted (IX(N,M) = 1) degrees of freedom are removed from the final set of equilibrium equations. Deleted degrees of freedom are fixed (points of reaction), and any loads applied in these degrees of freedom are ignored by the program. Nodes that are used for geometric reference only (i.e., nodes not assigned to any element) must have all six (6) degrees of freedom deleted Nodal degrees of freedom having undefined stiffness (such as rotations in an all TRUSS model, out-of-plane components in a two-dimensional planar model, etc.) should be deleted. Deletions have the beneficial effect of reducing the size of the set of equations that must be solved. The table below lists the types of degrees of freedom that are defined by each different element type. The table was prepared assuming that the element has general orientation in (X,Y,Z) space.

DEGREES OF FREEDOM WITH DEFINED STIFFNESS

ELEMENT TYPE	δX	δY	δZ	$\delta\theta_X$	$\delta\theta_Y$	$\delta\theta_Z$
1. TRUSS	x	x	x			
2. BEAM	x	x	x	x	x	x
3. MEMBRANE	x	x	x			
4. 2D QUADRILATERAL		x	x			
5. 3D BRICK	x	x	x			
6. PLATE, SHELL	x	x	x	x	x	x
7. BOUNDARY	x	x	x	x	x	x

III. NODAL POINT DATA (continued)

	DEGREES OF FREEDOM WITH DEFINED STIFFNESS					
ELEMENT TYPE	δX	δY	δZ	$\delta\theta_X$	$\delta\theta_Y$	$\delta\theta_Z$
8. THICK SHELL	x	x	x			
9. 3D/PIPE	x	x	x	x	x	x

Hence, for an all 3D BRICK model, only the X,Y,Z translations are defined at the node, and the number of equations can be cut in half by deleting the three (3) rotational components at every node. If a node is common to two or more different element types, then the non-trivial degrees of freedom are found by combination. For example, all six (6) components are possible at a node common to both BEAM and TRUSS elements; i.e., the BEAM governs.

A "master slave" option is allowed to model rigid links in the system. For this case, IX(N,M) = K means that the Mth degree of freedom at node "N" is "slave" to (dependent on) the same (Mth) degree of freedom at node "K"; node "K" is said to be the master node to which node N is slave. Note that no actual beam need to run from node K to node N, however the following restrictions hold:

(a) Node one (1) cannot be a master node; i.e., $K \neq 1$.
(b) Nodes "N" and "K" must be beam-only nodes; i.e., no other element type may be connected to either node N or K.
(c) A node "N" can be slave to only one master node "K"; multiple nodes, however, can be slave to the same master.
(d) If the beam from "N" to "K" is to be a rigid link arbitrarily oriented in the X,Y,Z space, then all six (6) degrees of freedom at node "N" must be made slaves to node "K"

Displacement/rotation components for slave degrees of freedom at node "N" are not recovered for printing; i.e., zeroes appear as output for slave degrees of freedom.

(4) When CT (Col. 1) is equal to the character "C", the values input in CC 36-65 are interpreted as the cylindrical (R,Y,θ) coordinates of node "N". Y is the axis of symmetry. R is the distance of a point from the Y-axis. The angle θ is measured clockwise from the positive Z-axis when looking in the positive Y direction. The cylindrical coordinate values are printed as entered on the card, but immediately after printing the

III. NODAL POINT DATA (continued)

global cartesian values are computed from the input entries. Note that boundary condition codes always refer to the the (X,Y,Z) system even if the node happens to be located with cylindrical coordinates.

(5) Nodal point cards need not be input in node-order sequence; eventually, however, all nodes in the integer set $\{1, NUMNP\}$ must be defined. Joint data for a series of nodes

$$\{N_1,\ N_1+1 \times KN_2,\ N_1+2 \times KN_2, \ldots, N_2\}$$

may be generated from information given on two (2) cards in sequence:

CARD 1 $\quad N_1, IX(N_1,1), \ldots, IX(N_1,6), X(N_1), \ldots, KN_1, T(N_1)/$

CARD 2 $\quad N_2, IX(N_2,1), \ldots, IX(N_2,6), X(N_2), \ldots, KN_2, T(N_2)/$

KN_2 is the mesh generation parameter given on the second card of a sequence. The first generated node is $N_1+1 \times KN_2$; the second generated node is $N_1+2 \times KN_2$, etc. Generation continues until node number $N_2 - KN_2$ is established. Note that the node difference $N_2 - N_1$ must be evenly divisible by KN_2. Intermediate nodes between N_1 and N_2 are located at equal intervals along the straight line between the two points. Boundary condition codes for the generated data are set equal to the values given on the first card. Node temperatures are found by linear interpolation between $T(N_1)$ and $T(N_2)$. Coordinate generation is <u>always</u> performed in the (X,Y,Z) system, and no generation is performed if KN_2 is zero (blank).

(6) Nodal temperatures describe the actual (physical) temperature distribution in the structure. Average element temperatures established from the nodal values are used to select material properties and to compute thermal strains in the model (static analysis only).

IV. ELEMENT DATA

TYPE 1 - THREE-DIMENSIONAL TRUSS ELEMENTS

Truss elements are identified by the number 1. Axial forces and stresses are calculated for each member. A uniform temperature change and inertia loads in three directions can be considered as the basic element load conditions. The truss elements are described by the following sequence of cards:

A. Control Card (3I5)

Columns	1 - 5	The number 1
	6 - 10	Total number of truss elements
	11 - 15	Number of material property cards

B. Material Property Cards (I5,5F10.0)

There need be as many of the following cards as are necessary to define the properties listed below for each element in the structure.

Columns	1 - 5	Material identification number
	6 - 15	Modulus of elasticity
	16 - 25	Coefficient of thermal expansion
	26 - 35	Mass density (used to calculate mass matrix)
	36 - 45	Cross-sectional area
	46 - 55	Weight density (used to calculate gravity loads)

C. Element Load Factors (4F10.0) Four cards

Three cards specifying the fraction of gravity (in each of the three global coordinate directions) to be added to each element load case.

Card 1: Multiplier of gravity load in the +X direction

Columns	1 - 10	Element load case A
	11 - 20	Element load case B
	21 - 30	Element load case C
	31 - 40	Element load case D

Card 2: As above for gravity in the +Y direction

Card 3: As above for gravity in the +Z direction

Card 4: This indicates the fraction of the thermal load to be added to each of the element load cases.

D. Element Data Cards (4I5,F10.0,I5)

One card per element in increasing numerical order starting with one.

Columns 1 - 5 Element number

IV. ELEMENT DATA (continued)

Columns	6 - 10	Node number I
	11 - 15	Node number J
	16 - 20	Material property number
	21 - 30	Reference temperature for zero stress
	31 - 35	Optional parameter k used for automatic generation of element data.

NOTES/

(1) If a series of elements exist such that the element number, N_i, is one greater than the previous element number (i.e. $N_i = N_{i-1} + 1$) and the nodal point number can be given by

$$I_i = I_{i-1} + k$$

$$J_i = J_{i-1} + k$$

then only the first element in the series need be provided. The element identification number and the temperature for the generated elements are set equal to the values on the first card. If k (given on the first card) is input as zero it is set to 1 by the program.

(2) The element temperature increase ΔT used to calculate thermal loads is given by

$$\Delta T = (T_i + T_j)/2.0 - T_r$$

where $(T_i + T_j)/2.0$ is the average of the nodal temperatures specified on the nodal point data cards for nodes i and j; and T_r is the zero stress reference temperature specified on the element card. For truss elements it is generally more convenient to set $T_i = T_j = 0.0$ such that $\Delta T = -T_r$ (note the minus sign). Other types of member loadings can be specified using an equivalent ΔT. If a truss member has an initial lack of fit by an amount d (positive if too long) then $\Delta T = d/(\alpha L)$. If an initial prestress force P (positive if tensile) is applied to the member ends that is released after the member is connected to the rest of the structure then $\Delta T = - P/(\alpha A E)$. In the above formulas A = cross section area, L = member length and α = coefficient of thermal expansion.

V. CONCENTRATED LOAD/MASS DATA (2I5,6F10.4)

notes	columns	variable	entry
(1)	1 - 5	N	Nodal point number
(2)	6 - 10	L	Structure load case number: GE.1: static analysis EQ 0; dynamic analysis
	11 - 20	FX(N,L)	X-direction force (or translational mass coefficient)
	21 - 30	FY(N,L)	Y-direction force (or translational mass coefficient)
	31 - 40	FZ(N,L)	Z-direction force (or translational mass coefficient)
	41 - 50	MX(N,L)	X-axis moment (or rotational inertia)
	51 - 60	MY(N,L)	Y-axis moment (or rotational inertia)
	61 - 70	MZ(N,L)	Z-axis moment (or rotational inertia)

NOTES:

(1) For a static analysis case (NDYN.EQ.0), one card is required for each nodal point ("N") having applied (non-zero) concentrated forces or moments. All structure load cases must be grouped together for the node ("N") before data is entered for the next (higher) node at which loads are applied. Only the structure load cases for which node N is loaded need be given, but the structure load case numbers ("L") which are referenced must be supplied in ascending order. Node loadings must be defined (input) in increasing node number order, but again, only those nodes actually loaded are required as input. The static loads defined in this section act on the structure exactly as input and are not scaled, factored, etc. by the element load case (A,B,C,D) multipliers (Section VI, below). Nodal forces arising from element loadings are combined (additively) with any concentrated loads given in this section. Applied force/moment vectors act on the structure, positive in the positive global directions. Only one card is allowed per node per load case.

For a dynamic analysis case (NDYN.EQ.1,2, 3 or 4), structure load cases have no meaning, but the program expects to read data in this section nonetheless. In place of concentrated loads, lumped mass coefficients for the nodal degrees of freedom may be input for any (or all) nodes. The mass matrix is automatically constructed by the program from element geometry and associated material densities; the mass coefficients read in this section are combined (additively) with the existing element-based lumped mass matrix. For mass input, a node may only be specified once, and the load case number ("L") must be zero (or blank).

. CONCENTRATED LOAD/MASS DATA (2I5,6F10.4) (continued)

The program terminates reading loads (or mass) data when a zero (or blank) node number ("N") is encountered; i.e., terminate this section of input with a blank card. For the special case of a static analysis with no concentrated loads applied, input only one (1) blank card in this section. Similarly, a dynamic analysis in which the mass matrix is not to be augmented by any entries in this section requires only one (1) blank card as input.

(2) For a static analysis, structure load case numbers range from "1" to the total number of load cases requested on the Master Control Card ("LL"); thus, $1 \le L \le LL$, NDYN.EQ.0. For a dynamic analysis, only zero (0) references are allowed; thus, L = 0, NDYN.EQ.1,2 3, or 4.

VI. ELEMENT LOAD MULTIPLIERS (4F10.0)

notes	columns	variable	entry
(1,2)	1 - 10	EM(1)	Multiplier for element load case A
	11 - 20	EM(2)	Multiplier for element load case B
	21 - 30	EM(3)	Multiplier for element load case C
	31 - 40	EM(4)	Multiplier for element load case D

NOTES/

(1) One card must be given for each static (NDYN.EQ.0) structure load case requested on the Master Control Card ("LL"). The cards must reference load case numbers in ascending order. The four (4) element load sets (A,B,C,D), if created during the processing of element data (Section IV, above), are combined with any concentrated loads specified in Section V for the structure load cases. For example, suppose an analysis case calls for seven (7) static structure loading conditions (i.e., LL = 7), then the program expects to read seven (7) cards in this section. Further, suppose card number three (3) in this section contains the entries:

[EM(1),EM(2),EM(3),EM(4)] = [-3.0,0.0,2.0,0.0]

Structure load case three (3) will then be constructed using 100% of any concentrated loads specified in Section V minus (-) 300% of the loads in element set A plus (+) 200% of the loads in element set C. Load sets B and D will not be applied in structure load case 3. Element load sets may be referenced any number of times in order to construct different structure loading conditions. Element-based loads (gravity, thermal, etc.) can only be applied to the structure by means of the data entries in this section.

(2) If this case calls for one of the dynamic analysis options, supply only one blank card in this section. If the job is a dynamic re-start case (NDYN.EQ.-2 or -3), skip this section.

Static analysis input is complete with this section. Begin a new data case with a new Heading Card (see Section I).

C. Output of sample SAP IV program

```
VAN DE GRAAFF GLASS METAL TRUSS TYPE BRIDGE.NORMAL LOADS,72800 LBS. CO.

C O N T R O L   I N F O R M A T I O N

    NUMBER OF NODAL POINTS        =    20
    NUMBER OF ELEMENT TYPES       =     1
    NUMBER OF LOAD CASES          =     1
    NUMBER OF FREQUENCIES         =     0
    ANALYSIS CODE (NDYN)          =     0
      EQ.0,  STATIC
      EQ.1,  MODAL EXTRACTION
      EQ.2,  FORCED RESPONSE
      EQ.3,  RESPONSE SPECTRUM
      EQ.4,  DIRECT INTEGRATION
    SOLUTION MODE (MODEX)         =     0
      EQ.0,  EXECUTION
      EQ.1,  DATA CHECK
    NUMBER OF SUBSPACE
    ITERATION VECTORS (NAD)       =     0
    EQUATIONS PER BLOCK           =     0
    TAPE10 SAVE FLAG (N10SV)      =     0

NODAL POINT INPUT DATA
```

NODE NUMBER	BOUNDARY CONDITION CODES X	Y	Z	XX	YY	ZZ	NODAL POINT COORDINATES X	Y	Z		T
1	1	0	1	1	1	1	.000	-420.000	.000	0	72.000
2	1	0	0	1	1	1	.000	-324.000	46.000	0	72.000
3	1	0	0	1	1	1	.000	-324.000	.000	0	72.000
4	1	0	0	1	1	1	.000	-228.000	46.000	0	72.000
5	1	0	0	1	1	1	.000	-228.000	.000	0	72.000
6	1	0	0	1	1	1	.000	-132.000	46.000	0	72.000
7	1	0	0	1	1	1	.000	-132.000	.000	0	72.000
8	1	0	0	1	1	1	.000	-36.000	46.000	0	72.000
9	1	0	0	1	1	1	.000	-36.000	.000	0	72.000
10	1	0	0	1	1	1	.000	.000	46.000	0	72.000
11	1	0	0	1	1	1	.000	.000	.000	0	72.000
12	1	0	0	1	1	1	.000	36.000	46.000	0	72.000
13	1	0	0	1	1	1	.000	36.000	.000	0	72.000
14	1	0	0	1	1	1	.000	132.000	46.000	0	72.000
15	1	0	0	1	1	1	.000	132.000	.000	0	72.000
16	1	0	0	1	1	1	.000	228.000	46.000	0	72.000
17	1	0	0	1	1	1	.000	228.000	.000	0	72.000
18	1	0	0	1	1	1	.000	324.000	46.000	0	72.000
19	1	0	0	1	1	1	.000	324.000	.000	0	72.000
20	1	1	1	1	1	1	.000	420.000	.000	0	72.000

GENERATED NODAL DATA

NODE NUMBER	BOUNDARY CONDITION CODES X	Y	Z	XX	YY	ZZ	NODAL POINT COORDINATES X	Y	Z	T
1	1	0	1	1	1	1	.000	-420.000	.000	72.000
2	1	0	0	1	1	1	.000	-324.000	46.000	72.000
3	1	0	0	1	1	1	.000	-324.000	.000	72.000
4	1	0	0	1	1	1	.000	-228.000	46.000	72.000
5	1	0	0	1	1	1	.000	-228.000	.000	72.000
6	1	0	0	1	1	1	.000	-132.000	46.000	72.000
7	1	0	0	1	1	1	.000	-132.000	.000	72.000
8	1	0	0	1	1	1	.000	-36.000	46.000	72.000
9	1	0	0	1	1	1	.000	-36.000	.000	72.000
10	1	0	0	1	1	1	.000	.000	46.000	72.000
11	1	0	0	1	1	1	.000	.000	.000	72.000
12	1	0	0	1	1	1	.000	36.000	46.000	72.000
13	1	0	0	1	1	1	.000	36.000	.000	72.000
14	1	0	0	1	1	1	.000	132.000	46.000	72.000
15	1	0	0	1	1	1	.000	132.000	.000	72.000
16	1	0	0	1	1	1	.000	228.000	46.000	72.000
17	1	0	0	1	1	1	.000	228.000	.000	72.000
18	1	0	0	1	1	1	.000	324.000	46.000	72.000
19	1	0	0	1	1	1	.000	324.000	.000	72.000
20	1	1	1	1	1	1	.000	420.000	.000	72.000

EQUATION NUMBERS

N	X	Y	Z	XX	YY	ZZ
1	0	1	0	0	0	0
2	0	2	3	0	0	0
3	0	4	5	0	0	0
4	0	6	7	0	0	0
5	0	8	9	0	0	0
6	0	10	11	0	0	0
7	0	12	13	0	0	0
8	0	14	15	0	0	0
9	0	16	17	0	0	0
10	0	18	19	0	0	0
11	0	20	21	0	0	0
12	0	22	23	0	0	0
13	0	24	25	0	0	0
14	0	26	27	0	0	0
15	0	28	29	0	0	0
16	0	30	31	0	0	0
17	0	32	33	0	0	0
18	0	34	35	0	0	0
19	0	36	37	0	0	0
20	0	0	0	0	0	0

N	I	J	TYPE	TEMP	BAND
1	1	2	3	72.00	3
2	1	3	1	72.00	5
3	3	2	7	72.00	4
4	2	4	2	72.00	6
5	3	4	3	72.00	4
6	3	5	2	72.00	6
7	5	4	7	72.00	4
8	4	6	1	72.00	6
9	5	6	3	72.00	4
10	5	7	2	72.00	6
11	7	6	7	72.00	4
12	6	8	1	72.00	6
13	7	8	4	72.00	4
14	7	9	2	72.00	6
15	9	8	7	72.00	4
16	8	10	6	72.00	6
17	9	10	5	72.00	4
18	9	11	6	72.00	6
19	11	10	5	72.00	4
20	10	12	6	72.00	6
21	10	13	5	72.00	8
22	11	13	6	72.00	6
23	13	12	7	72.00	4
24	12	14	1	72.00	6
25	12	15	4	72.00	8
26	13	15	2	72.00	6
27	15	14	7	72.00	4
28	14	16	1	72.00	6
29	14	17	3	72.00	8
30	15	17	2	72.00	6
31	17	16	7	72.00	4
32	16	18	2	72.00	6
33	16	19	3	72.00	8
34	17	19	2	72.00	6
35	19	18	7	72.00	4
36	18	20	3	72.00	2
37	19	20	1	72.00	2

E Q U A T I O N   P A R A M E T E R S

```
TOTAL NUMBER OF EQUATIONS            =    37
BANDWIDTH                            =     8
NUMBER OF EQUATIONS IN A BLOCK       =    37
NUMBER OF BLOCKS                     =     1
```

NUMBER OF TRUSS MEMBERS= 37
NUMBER OF DIFF. MEMBERS= 7

TYPE	E	ALPHA	DEN	AREA	AT
1	.5000000E 07	.1860000E-05	.2100000E-03	.6400000E 02	.8044000E-01
2	.5000000E 07	.1860000E-05	.2100000E-03	.4800000E 02	.8044000E-01
3	.5000000E 07	.1860000E-05	.2100000E-03	.3600000E 02	.8044000E-01
4	.5000000E 07	.1860000E-05	.2100000E-03	.2400000E 02	.8044000E-01
5	.3000000E 08	.6500000E-05	.7300000E-03	.1500000E 02	.2830000E 00
6	.3000000E 08	.6500000E-05	.7300000E-03	.1200000E 02	.2830000E 00
7	.3000000E 08	.6500000E-05	.7300000E-03	.9000000E 01	.2830000E 00

ELEMENT LOAD MULTIPLIERS

	A	B	C	D
X-DIR	.000000E 00	.000000E 00	.000000E 00	.000000E 00
Y-DIR	.000000E 00	.000000E 00	.000000E 00	.000000E 00
Z-DIR	-.100000E 01	-.100000E 01	-.100000E 01	-.100000E 01
TEMP	.000000E 00	.000000E 00	.000000E 00	.000000E 00

N O D A L   L O A D S   (S T A T I C)   O R   M A S S E S   (D Y N A M I C)

NODE NUMBER	LOAD CASE	X-AXIS FORCE	Y-AXIS FORCE	Z-AXIS FORCE	X-AXIS MOMENT	Y-AXIS MOMENT	Z-AXIS MOMENT
1	1	.00000E 00	.72800E 05	.00000E 00	.00000E 00	.00000E 00	.00000E 00
3	1	.00000E 00	.00000E 00	-.25000E 04	.00000E 00	.00000E 00	.00000E 00
5	1	.00000E 00	.00000E 00	-.25000E 04	.00000E 00	.00000E 00	.00000E 00
7	1	.00000E 00	.00000E 00	-.25000E 04	.00000E 00	.00000E 00	.00000E 00
9	1	.00000E 00	.00000E 00	-.25000E 04	.00000E 00	.00000E 00	.00000E 00
11	1	.00000E 00	.00000E 00	-.50000E 03	.00000E 00	.00000E 00	.00000E 00
12	1	.00000E 00	.10000E 04	.00000E 00	.00000E 00	.00000E 00	.00000E 00
13	1	.00000E 00	.00000E 00	-.25000E 04	.00000E 00	.00000E 00	.00000E 00
15	1	.00000E 00	.00000E 00	-.25000E 04	.00000E 00	.00000E 00	.00000E 00
17	1	.00000E 00	.00000E 00	-.25000E 04	.00000E 00	.00000E 00	.00000E 00
19	1	.00000E 00	.00000E 00	-.25000E 04	.00000E 00	.00000E 00	.00000E 00

ELEMENT LOAD MULTIPLIERS

STRUCTURE LOAD CASE	A	B	C	D
1	.000	.000	.000	.000

ODE DISPLACEMENTS/ROTATIONS

NODE MBER	LOAD CASE	X-TRANSLATION	Y-TRANSLATION	Z-TRANSLATION	X-ROTATION	Y-ROTATION	Z-ROTATION
20	1	.00000E 00	.00000E 00	.00000E 00	.00000E 00	.00000E 00	.00000E 00
19	1	.00000E 00	.15688E-01	-.14201E-01	.00000E 00	.00000E 00	.00000E 00
18	1	.00000E 00	.96749E-02	-.12446E-01	.00000E 00	.00000E 00	.00000E 00
17	1	.00000E 00	.30091E-01	-.34647E-01	.00000E 00	.00000E 00	.00000E 00
16	1	.00000E 00	.18277E-01	-.33518E-01	.00000E 00	.00000E 00	.00000E 00
15	1	.00000E 00	.40065E-01	-.53545E-01	.00000E 00	.00000E 00	.00000E 00
14	1	.00000E 00	.29615E-01	-.52641E-01	.00000E 00	.00000E 00	.00000E 00
13	1	.00000E 00	.47698E-01	-.58562E-01	.00000E 00	.00000E 00	.00000E 00
12	1	.00000E 00	.44275E-01	-.58084E-01	.00000E 00	.00000E 00	.00000E 00
11	1	.00000E 00	.49582E-01	-.57151E-01	.00000E 00	.00000E 00	.00000E 00

10	1	.00000E 00	.49646E-01	-.57100E-01	.00000E 00	.00000E 00	.00000E 00
9	1	.00000E 00	.51466E-01	-.58484E-01	.00000E 00	.00000E 00	.00000E 00
8	1	.00000E 00	.55027E-01	-.58025E-01	.00000E 00	.00000E 00	.00000E 00
7	1	.00000E 00	.59065E-01	-.53648E-01	.00000E 00	.00000E 00	.00000E 00
6	1	.00000E 00	.69481E-01	-.52762E-01	.00000E 00	.00000E 00	.00000E 00
5	1	.00000E 00	.68913E-01	-.35125E-01	.00000E 00	.00000E 00	.00000E 00
4	1	.00000E 00	.80682E-01	-.33814E-01	.00000E 00	.00000E 00	.00000E 00
3	1	.00000E 00	.83099E-01	-.14486E-01	.00000E 00	.00000E 00	.00000E 00
2	1	.00000E 00	.89192E-01	-.12749E-01	.00000E 00	.00000E 00	.00000E 00
1	1	.00000E 00	.98556E-01	.00000E 00	.00000E 00	.00000E 00	.00000E 00

TRUSS MEMBER ACTIONS

MEMBER	LOAD	STRESS	FORCE
1	1	-655.37568	-23593.525
2	1	-805.04658	-51522.981
3	1	1132.80423	10195.238
4	1	-443.27122	-21277.019
5	1	-494.66936	-17808.097
6	1	-738.81988	-35463.354
7	1	855.02646	7695.238
8	1	-583.38509	-37336.646
9	1	-333.96304	-12022.669
10	1	-512.93996	-24621.118
11	1	577.24868	5195.238
12	1	-752.79503	-48178.882
13	1	-259.88508	-6237.242
14	1	-395.75569	-18996.273
15	1	299.47090	2695.238
16	1	-4483.64389	-53803.727
17	1	-16.52799	-247.920
18	1	-1570.26986	-18843.478
19	1	33.33333	500.000

20	1	-4476.50104	-53718.012
21	1	-25.79979	-386.997
22	1	-1570.28986	-18843.478
23	1	311.64021	2604.762
24	1	-763.50932	-48864.596
25	1	-270.44578	-6490.699
26	1	-397.54141	-19081.988
27	1	589.41799	5304.762
28	1	-590.52795	-37793.789
29	1	-341.00351	-12276.126
30	1	-519.48758	-24935.404
31	1	867.19577	7804.762
32	1	-448.03313	-21505.590
33	1	-501.70983	-18061.554
34	1	-750.12940	-36006.211
35	1	1144.97354	10304.762
36	1	-662.41615	-23846.981
37	1	-817.10016	-52294.410

S T A T I C   S O L U T I O N   T I M E   L O G

```
EQUATION SOLUTION     =      .80
DISPLACEMENT OUTPUT   =      .00
STRESS RECOVERY       =     1.43
```

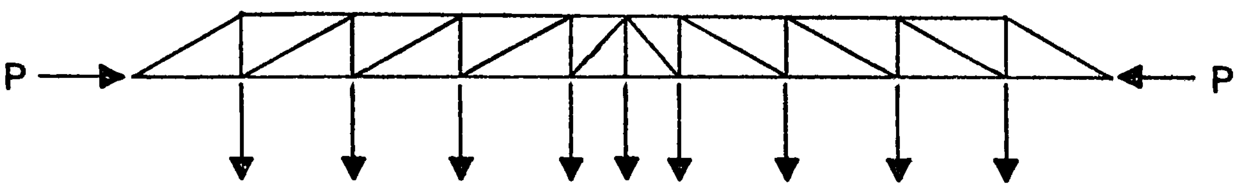
D. DISCUSSION OF SAP IV DISPLACEMENT SOLUTION.

The displacement at the midpoint of the truss was calculated by SAP IV to be .058". The exact solution, by the method of virtual complementary work, shows the displacement to be .091".

This difference can be explained as follows:

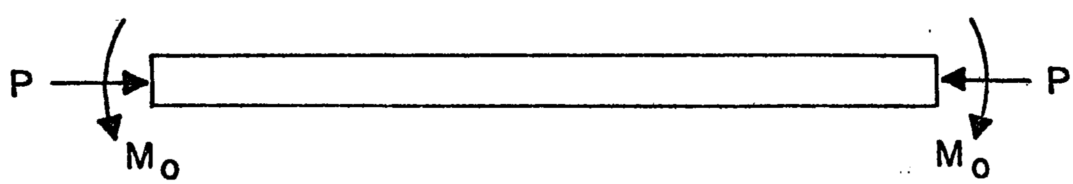
1. SAP IV solves for the truss-displacements by assuming the truss to be a simple beam.
2. SAP IV uses the principle of superposition.

The actual truss is loaded as shown:



By using simple beam theory and the superposition principle the truss can be idealized as:

a)



where: $P = 72\ 800$ lbs.

$E = 5 \times 10^6$ psi

$I = 144$ in.^{4}

The modulus of elasticity of the glass-metal truss-members is used since these are the members under compression.

The moment of inertia is also based on the cross-section of the glass-metal truss-members.

The displacement equation for this case is:

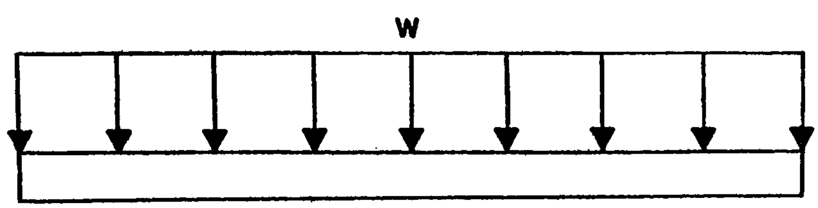
$$Y_{(x = \frac{1}{2}L)} = e\left(\sec \frac{kL}{2} - 1\right)$$

where $e = 22" =$ eccentricity of load P from central axis of beam.

$$k = \left(\frac{P}{EI}\right)^{1/2}$$

then: $$Y_{(x = \frac{1}{2}L)} = .059"$$

b)



where: $w = 24$ lbs./in.

$E = 30 \times 10^6$ psi

$I = 44\ 612$ in.^{4}

Since the transverse loads act only on the vertical steel truss-members the modulus of elasticity of steel is used. The moment of inertia is based on the cross-sectional area of the vertical steel members.

The displacement equation for this type of loading,is:

$$Y_{(x=\ 1/2\ L)} = \frac{5wL^4}{384\ EI} = -.116"$$

The total deflection is then:

$$Y_1 + Y_2 = .059" - .116" = -.057"$$

This total displacement agrees extremely well with the SAP IV solution.

The conclusion therefore is that the above assumptions are valid as far as SAP IV is concerned.

However,they do not give the correct results for displacements and should not be used for actual calculations.